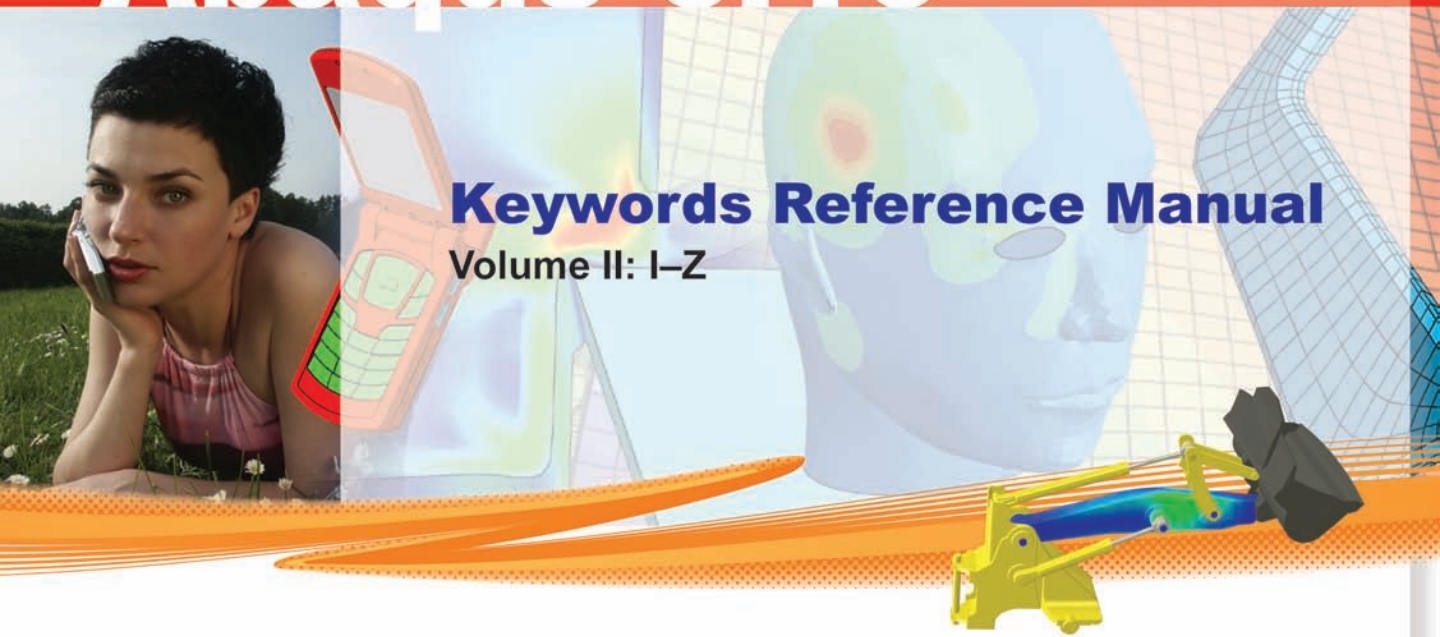
Abaqus 6.10

Keywords Reference Manual

Volume II: I–Z



SIMULIA

Abaqus Keywords

Reference Manual

Volume II

Legal Notices

CAUTION: This documentation is intended for qualified users who will exercise sound engineering judgment and expertise in the use of the Abaqus Software. The Abaqus Software is inherently complex, and the examples and procedures in this documentation are not intended to be exhaustive or to apply to any particular situation. Users are cautioned to satisfy themselves as to the accuracy and results of their analyses.

Dassault Systèmes and its subsidiaries, including Dassault Systèmes Simulia Corp., shall not be responsible for the accuracy or usefulness of any analysis performed using the Abaqus Software or the procedures, examples, or explanations in this documentation. Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

The Abaqus Software is available only under license from Dassault Systèmes or its subsidiary and may be used or reproduced only in accordance with the terms of such license. This documentation is subject to the terms and conditions of either the software license agreement signed by the parties, or, absent such an agreement, the then current software license agreement to which the documentation relates.

This documentation and the software described in this documentation are subject to change without prior notice.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiary.

The Abaqus Software is a product of Dassault Systèmes Simulia Corp., Providence, RI, USA.

© Dassault Systèmes, 2010

Abaqus, the 3DS logo, SIMULIA, CATIA, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the United States and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the Abaqus 6.10 Release Notes and the notices at: http://www.simulia.com/products/products_legal.html.

Locations

SIMULIA Worldwide Headquarters	Rising Sun Mills, 166 Valley Street, Providence, RI 02909–2499, Tel: +1 401 276 4400, Fax: +1 401 276 4408, simulia.support@3ds.com http://www.simulia.com
SIMULIA European Headquarters	Gaetano Martinolaan 95, P. O. Box 1637, 6201 BP Maastricht, The Netherlands, Tel: +31 43 356 6906, Fax: +31 43 356 6908, simulia.europe.info@3ds.com

Technical Support Centers

United States	Fremont, CA, Tel: +1 510 794 5891, simulia.west.support@3ds.com
	West Lafayette, IN, Tel: +1 765 497 1373, simulia.central.support@3ds.com
	Northville, MI, Tel: +1 248 349 4669, simulia.greatlakes.info@3ds.com
	Woodbury, MN, Tel: +1 612 424 9044, simulia.central.support@3ds.com
	Beachwood, OH, Tel: +1 216 378 1070, simulia.erie.info@3ds.com
	West Chester, OH, Tel: +1 513 275 1430, simulia.central.support@3ds.com
	Warwick, RI, Tel: +1 401 739 3637, simulia.east.support@3ds.com
	Lewisville, TX, Tel: +1 972 221 6500, simulia.south.info@3ds.com
Australia	Richmond VIC, Tel: +61 3 9421 2900, simulia.au.support@3ds.com
Austria	Vienna, Tel: +43 1 22 707 200, simulia.at.info@3ds.com
Benelux	Huizen, The Netherlands, Tel: +31 35 52 58 424, simulia.benelux.support@3ds.com
Canada	Toronto, ON, Tel: +1 416 402 2219, simulia.greatlakes.info@3ds.com
China	Beijing, P. R. China, Tel: +8610 6536 2288, simulia.cn.support@3ds.com
	Shanghai, P. R. China, Tel: +8621 3856 8000, simulia.cn.support@3ds.com
Czech & Slovak Republics	Synerma s. r. o., Psáry, Prague-West, Tel: +420 603 145 769, abaqus@synerma.cz
Finland	Vantaa, Tel: +358 46 712 2247, simulia.nordic.info@3ds.com
France	Velizy Villacoublay Cedex, Tel: +33 1 61 62 72 72, simulia.fr.support@3ds.com
Germany	Aachen, Tel: +49 241 474 01 0, simulia.de.info@3ds.com
	Munich, Tel: +49 89 543 48 77 0, simulia.de.info@3ds.com
Greece	3 Dimensional Data Systems, Crete, Tel: +30 2821040012, support@3dds.gr
India	Chennai, Tamil Nadu, Tel: +91 44 43443000, simulia.in.info@3ds.com
Israel	ADCOM, Givataim, Tel: +972 3 7325311, shmulik.keidar@adcomsim.co.il
Italy	Lainate MI, Tel: +39 02 39211211, simulia.ity.info@3ds.com
Japan	Tokyo, Tel: +81 3 5442 6300, simulia.tokyo.support@3ds.com
	Osaka, Tel: +81 6 4803 5020, simulia.osaka.support@3ds.com
	Yokohama-shi, Kanagawa, Tel: +81 45 470 9381, isight.jp.info@3ds.com
Korea	Mapo-Gu, Seoul, Tel: +82 2 785 6707/8, simulia.kr.info@3ds.com
Latin America	Puerto Madero, Buenos Aires, Tel: +54 11 4312 8700, Horacio.Burbridge@3ds.com
Malaysia	WorleyParsons Advanced Analysis, Kuala Lumpur, Tel: +603 2039 9000, abaqus.my@worleyparsons.com
New Zealand	Matrix Applied Computing Ltd., Auckland, Tel: +64 9 623 1223, abaqus-tech@matrix.co.nz
Poland	BudSoft Sp. z o.o., Poznań, Tel: +48 61 8508 466, info@budsoft.com.pl
Russia, Belarus & Ukraine	TESIS Ltd., Moscow, Tel: +7 495 612 44 22, info@tesis.com.ru
Scandinavia	Västerås, Sweden, Tel: +46 21 150870, simulia.nordic.info@3ds.com
Singapore	WorleyParsons Advanced Analysis, Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
South Africa	Finite Element Analysis Services (Pty) Ltd., Parklands, Tel: +27 21 556 6462, feas@feas.co.za
Spain & Portugal	Principia Ingenieros Consultores, S.A., Madrid, Tel: +34 91 209 1482, simulia@principia.es
Taiwan	Simutech Solution Corporation, Taipei, R.O.C., Tel: +886 2 2507 9550, lucille@simutech.com.tw
Thailand	WorleyParsons Advanced Analysis, Singapore, Tel: +65 6735 8444, abaqus.sg@worleyparsons.com
Turkey	A-Ztech Ltd., Istanbul, Tel: +90 216 361 8850, info@a-ztech.com.tr
United Kingdom	Warrington, Tel: +44 1 925 830900, simulia.uk.info@3ds.com
	Sevenoaks, Tel: +44 1 732 834930, simulia.uk.info@3ds.com

Complete contact information is available at http://www.simulia.com/locations/locations.html.

Preface

This section lists various resources that are available for help with using Abaqus Unified FEA software.

Support

Both technical engineering support (for problems with creating a model or performing an analysis) and systems support (for installation, licensing, and hardware-related problems) for Abaqus are offered through a network of local support offices. Regional contact information is listed in the front of each Abaqus manual and is accessible from the **Locations** page at www.simulia.com.

SIMULIA Online Support System

The SIMULIA Online Support System (SOSS) provides a knowledge database of SIMULIA Answers. The SIMULIA Answers are solutions to questions that we have had to answer or guidelines on how to use Abaqus, SIMULIA SLM, Isight, and other SIMULIA products. You can also submit new requests for support in the SOSS. All support incidents are tracked in the SOSS. If you are contacting us by means outside the SOSS to discuss an existing support problem and you know the incident number, please mention it so that we can consult the database to see what the latest action has been.

To use the SOSS, you need to register with the system. Visit the **My Support** page at www.simulia.com to register.

Many questions about Abaqus can also be answered by visiting the **Products** page and the **Support** page at www.simulia.com.

Anonymous ftp site

To facilitate data transfer with SIMULIA, an anonymous ftp account is available on the computer ftp.simulia.com. Login as user anonymous, and type your e-mail address as your password. Contact support before placing files on the site.

Training

All offices and representatives offer regularly scheduled public training classes. We also provide training seminars at customer sites. All training classes and seminars include workshops to provide as much practical experience with Abaqus as possible. For a schedule and descriptions of available classes, see www.simulia.com or call your local office or representative.

Feedback

We welcome any suggestions for improvements to Abaqus software, the support program, or documentation. We will ensure that any enhancement requests you make are considered for future releases. If you wish to make a suggestion about the service or products, refer to www.simulia.com. Complaints should be addressed by contacting your local office or through www.simulia.com by visiting the **Quality Assurance** section of the **Support** page.

Contents

Volume I

A

*ACOUSTIC FLOW VELOCITY 1.1
*ACOUSTIC MEDIUM 1.2
*ACOUSTIC WAVE FORMULATION 1.3
*ADAPTIVE MESH 1.4
*ADAPTIVE MESH CONSTRAINT 1.5
*ADAPTIVE MESH CONTROLS 1.6
*AMPLITUDE 1.7
*ANISOTROPIC HYPERELASTIC 1.8
*ANNEAL 1.9
*ANNEAL TEMPERATURE 1.10
*AQUA 1.11
*ASSEMBLY 1.12
*ASYMMETRIC-AXISYMMETRIC 1.13
*AXIAL 1.14

B

*BASE MOTION 2.1
*BASELINE CORRECTION 2.2
*BEAM ADDED INERTIA 2.3
*BEAM FLUID INERTIA 2.4
*BEAM GENERAL SECTION 2.5
*BEAM SECTION 2.6
*BEAM SECTION GENERATE 2.7
*BIAXIAL TEST DATA 2.8
*BLOCKAGE 2.9
*BOND 2.10
*BOUNDARY 2.11
*BRITTLE CRACKING 2.12
*BRITTLE FAILURE 2.13
*BRITTLE SHEAR 2.14
*BUCKLE 2.15
*BUCKLING ENVELOPE 2.16
*BUCKLING LENGTH 2.17
*BUCKLING REDUCTION FACTORS 2.18
*BULK VISCOSITY 2.19

C

*C ADDED MASS 3.1
*CAP CREEP 3.2
*CAP HARDENING 3.3
*CAP PLASTICITY 3.4
*CAPACITY 3.5
*CAST IRON COMPRESSION HARDENING 3.6
*CAST IRON PLASTICITY 3.7
*CAST IRON TENSION HARDENING 3.8
*CAVITY DEFINITION 3.9
*CECHARGE 3.10
*CECURRENT 3.11
*CENTROID 3.12
*CFILM 3.13
*CFLOW 3.14
*CFLUX 3.15
*CHANGE FRICTION 3.16
*CLAY HARDENING 3.17
*CLAY PLASTICITY 3.18
*CLEARANCE 3.19
*CLOAD 3.20
*COHESIVE BEHAVIOR 3.21
*COHESIVE SECTION 3.22
*COMBINED TEST DATA 3.23
*COMPLEX FREQUENCY 3.24
*CONCRETE 3.25
*CONCRETE COMPRESSION DAMAGE 3.26
*CONCRETE COMPRESSION HARDENING 3.27
*CONCRETE DAMAGED PLASTICITY 3.28
*CONCRETE TENSION DAMAGE 3.29
*CONCRETE TENSION STIFFENING 3.30
*CONDUCTIVITY 3.31
*CONNECTOR BEHAVIOR 3.32
*CONNECTOR CONSTITUTIVE REFERENCE 3.33
*CONNECTOR DAMAGE EVOLUTION 3.34
*CONNECTOR DAMAGE INITIATION 3.35
*CONNECTOR DAMPING 3.36
*CONNECTOR DERIVED COMPONENT 3.37
*CONNECTOR ELASTICITY 3.38
*CONNECTOR FAILURE 3.39
*CONNECTOR FRICTION 3.40

*CONNECTOR HARDENING 3.41
*CONNECTOR LOAD 3.42
*CONNECTOR LOCK 3.43
*CONNECTOR MOTION 3.44
*CONNECTOR PLASTICITY 3.45
*CONNECTOR POTENTIAL 3.46
*CONNECTOR SECTION 3.47
*CONNECTOR STOP 3.48
*CONNECTOR UNIAXIAL BEHAVIOR 3.49
*CONSTRAINT CONTROLS 3.50
*CONTACT 3.51
*CONTACT CLEARANCE 3.52
*CONTACT CLEARANCE ASSIGNMENT 3.53
*CONTACT CONTROLS 3.54
*CONTACT CONTROLS ASSIGNMENT 3.55
*CONTACT DAMPING 3.56
*CONTACT EXCLUSIONS 3.57
*CONTACT FILE 3.58
*CONTACT FORMULATION 3.59
*CONTACT INCLUSIONS 3.60
*CONTACT INITIALIZATION ASSIGNMENT 3.61
*CONTACT INITIALIZATION DATA 3.62
*CONTACT INTERFERENCE 3.63
*CONTACT OUTPUT 3.64
*CONTACT PAIR 3.65
*CONTACT PRINT 3.66
*CONTACT PROPERTY ASSIGNMENT 3.67
*CONTACT RESPONSE 3.68
*CONTACT STABILIZATION 3.69
*CONTOUR INTEGRAL 3.70
*CONTROLS 3.71
*CONWEP CHARGE PROPERTY 3.72
*CORRELATION 3.73
*CO-SIMULATION 3.74
*CO-SIMULATION CONTROLS 3.75
*CO-SIMULATION REGION 3.76
*COUPLED TEMPERATURE-DISPLACEMENT 3.77
*COUPLED THERMAL-ELECTRICAL 3.78
*COUPLING 3.79
*CRADIATE 3.80
*CREEP 3.81
*CREEP STRAIN RATE CONTROL 3.82

*CRUSHABLE FOAM 3.83
*CRUSHABLE FOAM HARDENING 3.84
*CYCLED PLASTIC 3.85
*CYCLIC 3.86
*CYCLIC HARDENING 3.87
*CYCLIC SYMMETRY MODEL 3.88

D

*D ADDED MASS 4.1
*DAMAGE EVOLUTION 4.2
*DAMAGE INITIATION 4.3
*DAMAGE STABILIZATION 4.4
*DAMPING 4.5
*DAMPING CONTROLS 4.6
*DASHPOT 4.7
*DEBOND 4.8
*DECHARGE 4.9
*DECURRENT 4.10
*DEFORMATION PLASTICITY 4.11
*DENSITY 4.12
*DEPVAR 4.13
*DESIGN GRADIENT 4.14
*DESIGN PARAMETER 4.15
*DESIGN RESPONSE 4.16
*DETONATION POINT 4.17
*DFLOW 4.18
*DFLUX 4.19
*DIAGNOSTICS 4.20
*DIELECTRIC 4.21
*DIFFUSIVITY 4.22
*DIRECT CYCLIC 4.23
*DISPLAY BODY 4.24
*DISTRIBUTING 4.25
*DISTRIBUTING COUPLING 4.26
*DISTRIBUTION 4.27
*DISTRIBUTION TABLE 4.28
*DLOAD 4.29
*DRAG CHAIN 4.30
*DRUCKER PRAGER 4.31
*DRUCKER PRAGER CREEP 4.32
*DRUCKER PRAGER HARDENING 4.33
*DSA CONTROLS 4.34
*DSECHARGE 4.35

*DSECURRENT 4.36
*DSFLOW 4.37
*DSFLUX 4.38
*DSLOAD 4.39
*DYNAMIC 4.40
*DYNAMIC TEMPERATURE-DISPLACEMENT 4.41

E

*EL FILE 5.1
*EL PRINT 5.2
*ELASTIC 5.3
*ELCOPY 5.4
*ELECTRICAL CONDUCTIVITY 5.5
*ELEMENT 5.6
*ELEMENT MATRIX OUTPUT 5.7
*ELEMENT OUTPUT 5.8
*ELEMENT RESPONSE 5.9
*ELGEN 5.10
*ELSET 5.11
*EMBEDDED ELEMENT 5.12
*EMISSIVITY 5.13
*END ASSEMBLY 5.14
*END INSTANCE 5.15
*END LOAD CASE 5.16
*END PART 5.17
*END STEP 5.18
*ENERGY FILE 5.19
*ENERGY OUTPUT 5.20
*ENERGY PRINT 5.21
*ENRICHMENT 5.22
*ENRICHMENT ACTIVATION 5.23
*EOS 5.24
*EOS COMPACTION 5.25
*EPJOINT 5.26
*EQUATION 5.27
*EULERIAN BOUNDARY 5.28
*EULERIAN MESH MOTION 5.29
*EULERIAN SECTION 5.30
*EXPANSION 5.31
*EXTREME ELEMENT VALUE 5.32
*EXTREME NODE VALUE 5.33
*EXTREME VALUE 5.34

F

*FABRIC 6.1
*FAIL STRAIN 6.2
*FAIL STRESS 6.3
*FAILURE RATIOS 6.4
*FASTENER 6.5
*FASTENER PROPERTY 6.6
*FIELD 6.7
*FILE FORMAT 6.8
*FILE OUTPUT 6.9
*FILM 6.10
*FILM PROPERTY 6.11
*FILTER 6.12
*FIXED MASS SCALING 6.13
*FLOW 6.14
*FLUID BEHAVIOR 6.15
*FLUID BULK MODULUS 6.16
*FLUID CAVITY 6.17
*FLUID DENSITY 6.18
*FLUID EXCHANGE 6.19
*FLUID EXCHANGE ACTIVATION 6.20
*FLUID EXCHANGE PROPERTY 6.21
*FLUID EXPANSION 6.22
*FLUID FLUX 6.23
*FLUID INFLATOR 6.24
*FLUID INFLATOR ACTIVATION 6.25
*FLUID INFLATOR MIXTURE 6.26
*FLUID INFLATOR PROPERTY 6.27
*FLUID LEAKOFF 6.28
*FLUID LINK 6.29
*FLUID PROPERTY 6.30
*FOUNDATION 6.31
*FRACTURE CRITERION 6.32
*FRAME SECTION 6.33
*FREQUENCY 6.34
*FRICTION 6.35

G

*GAP 7.1
*GAP CONDUCTANCE 7.2
*GAP ELECTRICAL CONDUCTANCE 7.3

*GAP FLOW 7.4
*GAP HEAT GENERATION 7.5
*GAP RADIATION 7.6
*GASKET BEHAVIOR 7.7
*GASKET CONTACT AREA 7.8
*GASKET ELASTICITY 7.9
*GASKET SECTION 7.10
*GASKET THICKNESS BEHAVIOR 7.11
*GAS SPECIFIC HEAT 7.12
*GEL 7.13
*GEOSTATIC 7.14
*GLOBAL DAMPING 7.15

H

*HEADING 8.1
*HEAT GENERATION 8.2
*HEAT TRANSFER 8.3
*HEATCAP 8.4
*HOURGLASS STIFFNESS 8.5
*HYPERELASTIC 8.6
*HYPERFOAM 8.7
*HYPOELASTIC 8.8
*HYSTERESIS 8.9

Volume II

I

*IMPEDANCE 9.1
*IMPEDANCE PROPERTY 9.2
*IMPERFECTION 9.3
*IMPORT 9.4
*IMPORT CONTROLS 9.5
*IMPORT ELSET 9.6
*IMPORT NSET 9.7
*INCIDENT WAVE 9.8
*INCIDENT WAVE FLUID PROPERTY 9.9
*INCIDENT WAVE INTERACTION 9.10
*INCIDENT WAVE INTERACTION PROPERTY 9.11
*INCIDENT WAVE PROPERTY 9.12
*INCIDENT WAVE REFLECTION 9.13
*INCLUDE 9.14
*INCREMENTATION OUTPUT 9.15
*INELASTIC HEAT FRACTION 9.16
*INERTIA RELIEF 9.17
*INITIAL CONDITIONS 9.18
*INSTANCE 9.19
*INTEGRATED OUTPUT 9.20
*INTEGRATED OUTPUT SECTION 9.21
*INTERFACE 9.22
*ITS 9.23

J

*JOINT 10.1
*JOINT ELASTICITY 10.2
*JOINT PLASTICITY 10.3
*JOINTED MATERIAL 10.4
*JOULE HEAT FRACTION 10.5

K

*KAPPA 11.1
*KINEMATIC 11.2
*KINEMATIC COUPLING 11.3

L

*LATENT HEAT 12.1
*LOAD CASE 12.2
*LOADING DATA 12.3
*LOW DENSITY FOAM 12.4

M

*MAP SOLUTION 13.1
*MASS 13.2
*MASS DIFFUSION 13.3
*MASS FLOW RATE 13.4
*MATERIAL 13.5
*MATRIX 13.6
*MATRIX ASSEMBLE 13.7
*MATRIX GENERATE 13.8
*MATRIX INPUT 13.9
*MEMBRANE SECTION 13.10
*MODAL DAMPING 13.11
*MODAL DYNAMIC 13.12
*MODAL FILE 13.13
*MODAL OUTPUT 13.14
*MODAL PRINT 13.15
*MODEL CHANGE 13.16
*MOHR COULOMB 13.17
*MOHR COULOMB HARDENING 13.18
*MOISTURE SWELLING 13.19
*MOLECULAR WEIGHT 13.20
*MONITOR 13.21
*MOTION 13.22
*MPC 13.23
*MULLINS EFFECT 13.24
*M1 13.25
*M2 13.26

N

*NCOPY 14.1
*NFILL 14.2
*NGEN 14.3
*NMAP 14.4
*NO COMPRESSION 14.5

*NO TENSION 14.6
*NODAL ENERGY RATE 14.7
*NODAL THICKNESS 14.8
*NODE 14.9
*NODE FILE 14.10
*NODE OUTPUT 14.11
*NODE PRINT 14.12
*NODE RESPONSE 14.13
*NONSTRUCTURAL MASS 14.14
*NORMAL 14.15
*NSET 14.16

O

*ORIENTATION 15.1
*ORNL 15.2
*OUTPUT 15.3

P, Q

*PARAMETER 16.1
*PARAMETER DEPENDENCE 16.2
*PARAMETER SHAPE VARIATION 16.3
*PART 16.4
*PERIODIC 16.5
*PERMEABILITY 16.6
*PHYSICAL CONSTANTS 16.7
*PIEZOELECTRIC 16.8
*PIPE-SOIL INTERACTION 16.9
*PIPE-SOIL STIFFNESS 16.10
*PLANAR TEST DATA 16.11
*PLASTIC 16.12
*PLASTIC AXIAL 16.13
*PLASTIC M1 16.14
*PLASTIC M2 16.15
*PLASTIC TORQUE 16.16
*POROUS BULK MODULI 16.17
*POROUS ELASTIC 16.18
*POROUS FAILURE CRITERIA 16.19
*POROUS METAL PLASTICITY 16.20
*POST OUTPUT 16.21
*POTENTIAL 16.22
*PREPRINT 16.23
*PRESSURE PENETRATION 16.24

*PRESSURE STRESS 16.25
*PRESTRESS HOLD 16.26
*PRE-TENSION SECTION 16.27
*PRINT 16.28
*PSD-DEFINITION 16.29

R

*RADIATE 17.1
*RADIATION FILE 17.2
*RADIATION OUTPUT 17.3
*RADIATION PRINT 17.4
*RADIATION SYMMETRY 17.5
*RADIATION VIEWFACTOR 17.6
*RANDOM RESPONSE 17.7
*RATE DEPENDENT 17.8
*RATIOS 17.9
*REACTION RATE 17.10
*REBAR 17.11
*REBAR LAYER 17.12
*REFLECTION 17.13
*RELEASE 17.14
*RESPONSE SPECTRUM 17.15
*RESTART 17.16
*RETAINED NODAL DOFS 17.17
*RIGID BODY 17.18
*RIGID SURFACE 17.19
*ROTARY INERTIA 17.20

S

*SECTION CONTROLS 18.1
*SECTION FILE 18.2
*SECTION ORIGIN 18.3
*SECTION POINTS 18.4
*SECTION PRINT 18.5
*SELECT CYCLIC SYMMETRY MODES 18.6
*SELECT EIGENMODES 18.7
*SFILM 18.8
*SFLOW 18.9
*SHEAR CENTER 18.10
*SHEAR FAILURE 18.11
*SHEAR RETENTION 18.12
*SHEAR TEST DATA 18.13

*SHELL GENERAL SECTION 18.14
*SHELL SECTION 18.15
*SHELL TO SOLID COUPLING 18.16
*SIMPEDANCE 18.17
*SIMPLE SHEAR TEST DATA 18.18
*SLIDE LINE 18.19
*SLOAD 18.20
*SOILS 18.21
*SOLID SECTION 18.22
*SOLUBILITY 18.23
*SOLUTION TECHNIQUE 18.24
*SOLVER CONTROLS 18.25
*SORPTION 18.26
*SPECIFIC HEAT 18.27
*SPECTRUM 18.28
*SPRING 18.29
*SRADIATE 18.30
*STATIC 18.31
*STEADY STATE CRITERIA 18.32
*STEADY STATE DETECTION 18.33
*STEADY STATE DYNAMICS 18.34
*STEADY STATE TRANSPORT 18.35
*STEP 18.36
*SUBCYCLING 18.37
*SUBMODEL 18.38
*SUBSTRUCTURE COPY 18.39
*SUBSTRUCTURE DELETE 18.40
*SUBSTRUCTURE DIRECTORY 18.41
*SUBSTRUCTURE GENERATE 18.42
*SUBSTRUCTURE LOAD CASE 18.43
*SUBSTRUCTURE MATRIX OUTPUT 18.44
*SUBSTRUCTURE PATH 18.45
*SUBSTRUCTURE PROPERTY 18.46
*SURFACE 18.47
*SURFACE BEHAVIOR 18.48
*SURFACE FLAW 18.49
*SURFACE INTERACTION 18.50
*SURFACE PROPERTY 18.51
*SURFACE PROPERTY ASSIGNMENT 18.52
*SURFACE SECTION 18.53
*SURFACE SMOOTHING 18.54
*SWELLING 18.55

*SYMMETRIC MODEL GENERATION 18.56
*SYMMETRIC RESULTS TRANSFER 18.57
*SYSTEM 18.58

T

*TEMPERATURE 19.1
*TENSILE FAILURE 19.2
*TENSION CUTOFF 19.3
*TENSION STIFFENING 19.4
*THERMAL EXPANSION 19.5
*TIE 19.6
*TIME POINTS 19.7
*TORQUE 19.8
*TORQUE PRINT 19.9
*TRACER PARTICLE 19.10
*TRANSFORM 19.11
*TRANSPORT VELOCITY 19.12
*TRANSVERSE SHEAR STIFFNESS 19.13
*TRIAXIAL TEST DATA 19.14
*TRS 19.15

U

*UEL PROPERTY 20.1
*UNDEX CHARGE PROPERTY 20.2
*UNIAXIAL 20.3
*UNIAXIAL TEST DATA 20.4
*UNLOADING DATA 20.5
*USER DEFINED FIELD 20.6
*USER ELEMENT 20.7
*USER MATERIAL 20.8
*USER OUTPUT VARIABLES 20.9

V

*VARIABLE MASS SCALING 21.1
*VIEWFACTOR OUTPUT 21.2
*VISCO 21.3
*VISCOSITY 21.4
*VISCOELASTIC 21.5
*VISCOUS 21.6
*VOID NUCLEATION 21.7
*VOLUMETRIC TEST DATA 21.8

W, X, Y, Z

*WAVE 22.1
*WIND 22.2

9. I

9.1 *IMPEDANCE: Define impedances for acoustic analysis.

This option is used to provide boundary impedances or nonreflecting boundaries for acoustic and coupled acoustic-structural analyses.

Products: Abaqus/Standard Abaqus/Explicit

Type: History data

Level: Step

References:

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Manual
- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *IMPEDANCE PROPERTY
- *SIMPEDANCE

Required, mutually exclusive parameters:

PROPERTY

Set this parameter equal to the name of the *IMPEDANCE PROPERTY option defining the table of impedance values to be used.

NONREFLECTING

Set NONREFLECTING=PLANAR (default) to specify the impedance corresponding to that of a normal incidence plane wave.

Set NONREFLECTING=IMPROVED to specify the impedance corresponding to that of a plane wave at an arbitrary angle of incidence. This parameter can be used only for transient dynamics.

Set NONREFLECTING=CIRCULAR to specify a radiation condition appropriate for a circular boundary in two dimensions or a right circular cylinder in three dimensions.

Set NONREFLECTING=SPHERICAL to specify a radiation condition appropriate for a spherical boundary.

Set NONREFLECTING=ELLIPTICAL to specify a radiation condition appropriate for an elliptical boundary in two dimensions or a right elliptical cylinder in three dimensions.

Set NONREFLECTING=PROLATE SPHEROIDAL to specify a radiation condition appropriate for a prolate spheroidal boundary.

Optional parameter:

OP

Set OP=MOD (default) to modify existing impedances or to define additional impedances.

Set OP=NEW if all existing impedances applied to the model should be removed. To remove only selected impedances, use OP=NEW and respecify all impedances that are to be retained.

Data line to define an impedance for PROPERTY, NONREFLECTING=PLANAR, or NONREFLECTING=IMPROVED:

First (and only) line:

1. Element number or element set label.
2. Surface impedance type label, I*n*, for impedance on face *n*.

Data line to define an absorbing boundary impedance for NONREFLECTING=CIRCULAR or NONREFLECTING=SPHERICAL:

First (and only) line:

1. Element number or element set label.
2. Surface impedance type label, I*n*, for impedance on face *n*.
3. r_1, the radius of the circle or sphere defining the absorbing boundary surface.

Data line to define an absorbing boundary impedance for NONREFLECTING=ELLIPTICAL or NONREFLECTING=PROLATE SPHEROIDAL:

First (and only) line:

1. Element number or element set label.
2. Surface impedance type label, I*n*, for impedance on face *n*.
3. The semimajor axis, a, of the ellipse or prolate spheroid defining the surface. a is 1/2 of the maximum distance between two points on the ellipse or spheroid, analogous to the radius of a circle or sphere.
4. The eccentricity, ϵ, of the ellipse or prolate spheroid. The eccentricity is the square root of one minus the square of the ratio of the semiminor axis, b, to the semimajor axis, a: $\epsilon = \sqrt{1-(b/a)^2}$.
5. Global *X*-coordinate of the center of the ellipse or prolate spheroid defining the radiating surface.
6. Global *Y*-coordinate of the center of the ellipse or prolate spheroid defining the radiating surface.
7. Global *Z*-coordinate of the center of the ellipse or prolate spheroid defining the radiating surface.

8. *X*-component of the direction cosine of the major axis of the ellipse or prolate spheroid defining the radiating surface. The components of this vector need not be normalized to unit magnitude.
9. *Y*-component of the direction cosine of the major axis of the ellipse or prolate spheroid defining the radiating surface.
10. *Z*-component of the direction cosine of the major axis of the ellipse or prolate spheroid defining the radiating surface.

9.2 *IMPEDANCE PROPERTY: Define the impedance parameters for an acoustic medium boundary.

This option is used to define the proportionality factors between the pressure and the normal components of surface displacement and velocity in acoustic analysis. The *IMPEDANCE PROPERTY option must be used in conjunction with the *IMPEDANCE or *SIMPEDANCE option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *IMPEDANCE
- *SIMPEDANCE

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the impedance property on the *IMPEDANCE or *SIMPEDANCE option.

Optional parameters:

DATA

Set DATA=ADMITTANCE (default) to specify an impedance using a table of admittance values.

Set DATA=IMPEDANCE to specify an impedance using a table of real and imaginary parts of the impedance.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define an impedance using DATA=ADMITTANCE (default):

First line:

1. $1/k_1$, the proportionality factor between pressure and displacement of the surface in the normal direction. This quantity is the imaginary part of the complex admittance, divided by the angular frequency; see "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual. (Units of $F^{-1}L^3$.)
2. $1/c_1$, the proportionality factor between pressure and velocity of the surface in the normal direction. This quantity is the real part of the complex admittance. (Units of $F^{-1}L^3T^{-1}$.)
3. Frequency. (Cycles/time.) Frequency dependence is active only during frequency domain analysis in Abaqus/Standard.

Repeat this data line as often as necessary in Abaqus/Standard to describe the variation of the coefficients with frequency. Only the first line entered will be used in direct-integration procedures.

Data lines to define an impedance using DATA=IMPEDANCE:

First line:

1. $\mathrm{Re(Z)}$, the real part of the surface impedance. (Units of F L^{-3}T.)
2. $\mathrm{Im(Z)}$, the imaginary part of the surface impedance. (Units of F L^{-3}T.)
3. Frequency. (Cycles/time.) Frequency dependence is active only during frequency domain analysis in Abaqus/Standard.

Repeat this data line as often as necessary in Abaqus/Standard to describe the variation of the coefficients with frequency. Only the first line entered will be used in direct-integration procedures.

9.3 *IMPERFECTION: Introduce geometric imperfections for postbuckling analysis.

This option is used to introduce a geometric imperfection into a model for a postbuckling analysis.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Introducing a geometric imperfection into a model," Section 11.3.1 of the Abaqus Analysis User's Manual
- "Unstable collapse and postbuckling analysis," Section 6.2.4 of the Abaqus Analysis User's Manual
- "Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Manual

Optional parameters (mutually exclusive—if neither parameter is specified, Abaqus assumes that the imperfection data will be entered directly on the data lines):

FILE

Set this parameter equal to the name of the results file from a previous Abaqus/Standard analysis containing either the mode shapes from a *BUCKLE or *FREQUENCY analysis or the nodal displacements from a *STATIC analysis.

INPUT

Set this parameter equal to the name of the alternate input file containing the imperfection data, in general, as the node number and imperfection values in the global coordinate system. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names.

Required parameter if the FILE parameter is used:

STEP

Set this parameter equal to the step number (in the analysis whose results file is being used as input to this option) from which the modal or displacement data are to be read.

Optional parameters if the FILE parameter is used:

INC

Set this parameter equal to the increment number (in the analysis whose results file is being used as input to this option) from which the displacement data are to be read. If this parameter is omitted, Abaqus will read the data from the last increment available for the specified step on the results file.

NSET

Set this parameter equal to the node set to which the geometric imperfection values are to be applied. If this parameter is omitted, the imperfection will be applied to all nodes in the model.

Optional parameter if the FILE parameter is omitted:

SYSTEM

Set SYSTEM=R (default) to specify the imperfection as perturbation values of Cartesian coordinates. Set SYSTEM=C to specify the imperfection as perturbation values of cylindrical coordinates. Set SYSTEM=S to specify the imperfection as perturbation values of spherical coordinates. See Figure 9.3–1.

The SYSTEM parameter is entirely local to this option and should not be confused with the *SYSTEM option. As the data lines are read, the imperfection values specified are transformed to the global rectangular Cartesian coordinate system. This transformation requires that the object be centered about the origin of the global coordinate system; i.e., the *SYSTEM option should be off when specifying imperfections as perturbation values using either cylindrical or spherical coordinates.

Data lines to define the imperfection as a linear superposition of mode shapes from the results file:

First line:

1. Mode number.
2. Scaling factor for this mode.

Repeat this data line as often as necessary to define the imperfection as a linear combination of mode shapes.

Data line to define the imperfection based on the solution of a static analysis from the results file:

First (and only) line:

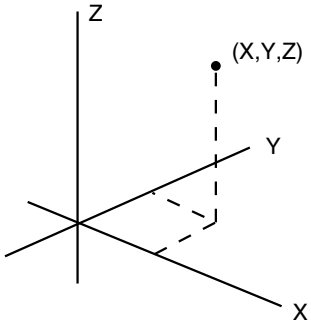
1. Set to 1.
2. Scaling factor.

Data lines to define the imperfection if the FILE and INPUT parameters are omitted:

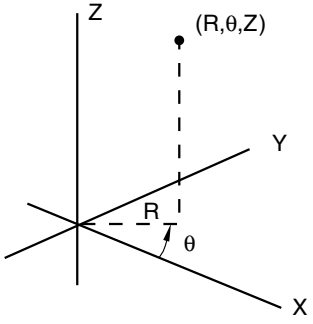
First line:

1. Node number.
2. Component of imperfection in the first coordinate direction.
3. Component of imperfection in the second coordinate direction.
4. Component of imperfection in the third coordinate direction.

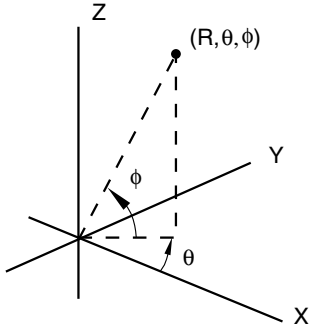
Repeat this data line as often as necessary to define the imperfection.



Rectangular Cartesian
(SYSTEM=R)
(default)



Cylindrical
(SYSTEM=C)



Spherical
(SYSTEM=S)

(θ and ϕ are given in degrees)

Figure 9.3–1 Coordinate systems.

9.4 *IMPORT: Import information from a previous Abaqus/Explicit or Abaqus/Standard analysis.

This option is used to define the time in a previous Abaqus/Standard or Abaqus/Explicit analysis at which the specified node and element information is imported. The *IMPORT option must be used in conjunction with the *INSTANCE option when importing a part instance from a previous analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part instance

Abaqus/CAE: Supported for use in conjunction with part instances; importing selected part instances stored in an output database is supported using the **File** menu and importing the initial state of part instances is supported in the Load module.

References:

- "Transferring results between Abaqus analyses: overview," Section 9.2.1 of the Abaqus Analysis User's Manual
- *INSTANCE

Required parameter:

UPDATE

Set UPDATE=NO to continue the analysis without resetting the reference configuration.

Set UPDATE=YES to continue the analysis by resetting the reference configuration to be the imported configuration. In this case displacement and strain values are calculated from the new reference configuration.

Optional, mutually exclusive parameters:

INCREMENT

When importing an analysis from Abaqus/Standard into Abaqus/Explicit or from one Abaqus/Standard analysis into another Abaqus/Standard analysis, set this parameter equal to the increment of the specified step on the Abaqus/Standard restart file from which the analysis is to be imported. If this parameter is omitted, the analysis is imported from the last available increment of the specified step.

INTERVAL

When importing an analysis from Abaqus/Explicit into Abaqus/Standard or from one Abaqus/Explicit analysis into another Abaqus/Explicit analysis, set this parameter equal to

the interval of the specified step on the Abaqus/Explicit state file from which the analysis is to be imported. If this parameter is omitted, the analysis is imported from the last available interval of the specified step.

ITERATION

This parameter is relevant only when the results are imported from a previous direct cyclic Abaqus/Standard analysis.

When importing an analysis from Abaqus/Standard into Abaqus/Explicit or from one Abaqus/Standard analysis into another Abaqus/Standard analysis, set this parameter equal to the iteration number of the specified step on the Abaqus/Standard restart file from which the analysis is to be imported. Since restart information can be written only at the end of an iteration in a direct cyclic analysis, the INCREMENT parameter is irrelevant and is ignored if the ITERATION parameter is specified. If this parameter is omitted, the analysis is imported from the last available iteration of the specified step.

Optional parameters:

STATE

Set STATE=YES (default) to import the current material state of the elements at the specified step and the specified interval, increment, or iteration.

Set STATE=NO if no material state is to be imported. In this case the elements will start with no initial state or with the state as defined by the *INITIAL CONDITIONS option.

STEP

Set this parameter equal to the step on the Abaqus/Explicit state file or on the Abaqus/Standard restart file from which the analysis is being imported. If this parameter is omitted, the analysis is imported from the last available step on the state file or the restart file at the specified increment, interval, or iteration.

Data lines to specify element sets to be imported:

First line:

1. List of element sets that are to be imported. Specify only element set names that are used to define the section properties in the previous Abaqus/Explicit or Abaqus/Standard analysis.

Repeat this data line as often as necessary to define the element sets to be imported. Up to 16 element sets can be listed per data line.

There are no data lines for importing a part instance.

9.5 *IMPORT CONTROLS: Specify tolerances used in importing model and results data.

This option is used to specify the tolerance for error checking on shell normals in Abaqus/Standard or Abaqus/Explicit when the *IMPORT, UPDATE=YES option is used. If the *IMPORT CONTROLS option is used, it must appear after the *IMPORT option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Transferring results between Abaqus/Explicit and Abaqus/Standard," Section 9.2.2 of the Abaqus Analysis User's Manual
- *IMPORT

Required parameter:

NORMAL TOL

Set this parameter equal to the tolerance required for the error checking on shell normals. The default value is 0.1.

There are no data lines associated with this option.

9.6 *IMPORT ELSET: Import element set definitions from a previous Abaqus/Explicit or Abaqus/Standard analysis.

This option is used to import element set definitions that were defined in a previous Abaqus/Explicit or Abaqus/Standard analysis. If the *IMPORT ELSET option is used, it must appear after the *IMPORT option. If this option is omitted or is specified without any data lines, all the element sets relevant to the analysis will be imported.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Transferring results between Abaqus/Explicit and Abaqus/Standard," Section 9.2.2 of the Abaqus Analysis User's Manual
- *IMPORT

There are no parameters associated with this option.

Data lines to specify element set definitions to be imported:

First line:

1. List of element set names for which the element set definitions are to be imported.

Repeat this data line as often as necessary to specify the element set definitions to be imported. Up to 16 element sets can be listed per line.

9.7 *IMPORT NSET: Import node set definitions from a previous Abaqus/Explicit or Abaqus/Standard analysis.

This option is used to import node set definitions that were defined in a previous Abaqus/Explicit or Abaqus/Standard analysis. If the *IMPORT NSET option is used, it must appear after the *IMPORT option. If this option is omitted or is specified without any data lines, all the node sets relevant to the analysis will be imported.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Transferring results between Abaqus/Explicit and Abaqus/Standard," Section 9.2.2 of the Abaqus Analysis User's Manual
- *IMPORT

There are no parameters associated with this option.

Data lines to specify node set definitions to be imported:

First line:

1. List of node set names for which the node set definitions are to be imported.

Repeat this data line as often as necessary to specify the node set definitions to be imported. Up to 16 node sets can be listed per line.

9.8 *INCIDENT WAVE: Define incident wave loading for a blast or scattering load on a boundary.

The preferred interface for applying incident wave loading is the *INCIDENT WAVE INTERACTION option used in conjunction with the *INCIDENT WAVE INTERACTION PROPERTY option. The alternative interface uses the *INCIDENT WAVE option to apply incident wave loading.

The *INCIDENT WAVE PROPERTY option must be used in conjunction with the *INCIDENT WAVE option. If the incident wave field includes a reflection off a plane outside the boundaries of the mesh, this effect can be modeled with the *INCIDENT WAVE REFLECTION option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; this option has been superseded by incident wave interactions.

References:

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Manual
- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *INCIDENT WAVE PROPERTY
- *INCIDENT WAVE REFLECTION

Required parameter:

PROPERTY

Set this parameter equal to the name of the *INCIDENT WAVE PROPERTY option defining the incident wave field.

Required, mutually exclusive parameters:

ACCELERATION AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining the fluid particle acceleration time history at the standoff point ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual). This amplitude curve will be used to compute the fluid traction *only*: a solid surface requiring a pressure load cannot be specified on the data line of an *INCIDENT WAVE option if the ACCELERATION AMPLITUDE parameter is used.

This parameter is valid only for planar incident waves using the *INCIDENT WAVE PROPERTY, TYPE=PLANE option. Reflected loads, using the *INCIDENT WAVE REFLECTION option, are not permitted in this case.

PRESSURE AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining the fluid pressure time history at the standoff point ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual). The corresponding fluid traction, if required, will be computed from the pressure amplitude reference.

Data lines to define an incident wave:

First line:

1. Surface name.
2. Reference magnitude.

Repeat this data line as often as necessary to describe the loading on the surfaces due to the incident wave. In problems involving fluid-solid boundaries, both the fluid surface and the solid surface comprising the boundary must have an incident wave load specified, using the appropriate load type.

9.9 *INCIDENT WAVE FLUID PROPERTY: Define the fluid properties associated with an incident wave.

The preferred interface for defining the fluid properties for an incident wave is the *INCIDENT WAVE INTERACTION PROPERTY option used in conjunction with the *INCIDENT WAVE INTERACTION option. The alternative interface uses the *INCIDENT WAVE FLUID PROPERTY option to define the fluid properties used to define an incident wave.

The *INCIDENT WAVE FLUID PROPERTY must be used in conjunction with the *INCIDENT WAVE PROPERTY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Unsupported; this option has been superseded by incident wave interaction properties.

References:

- "Acoustic medium," Section 23.3.1 of the Abaqus Analysis User's Manual
- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *INCIDENT WAVE PROPERTY

There are no parameters associated with this option.

Data line to define incident wave fluid properties:

First (and only) line:

1. K_f, the fluid bulk modulus.
2. ρ_f, the fluid mass density.

9.10 *INCIDENT WAVE INTERACTION: Define incident wave loading for a blast or scattering load on a surface.

This option is used to apply incident wave loading. The *INCIDENT WAVE INTERACTION PROPERTY option must be used in conjunction with the *INCIDENT WAVE INTERACTION option. If the incident wave field includes a reflection off a plane outside the boundaries of the mesh, this effect can be modeled with the *INCIDENT WAVE REFLECTION option. The incident wave interaction can be used in steady-state dynamic procedures to define individual wave sources or loading from diffuse sources.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Manual
- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *INCIDENT WAVE INTERACTION PROPERTY
- *INCIDENT WAVE REFLECTION

Required parameter:

PROPERTY

Set this parameter equal to the name of the *INCIDENT WAVE INTERACTION PROPERTY option defining the incident wave field.

Required, mutually exclusive parameters:

ACCELERATION AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining the fluid particle acceleration time history at the standoff point ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual). This amplitude curve will be used to compute the fluid traction *only*: a solid surface requiring a pressure load cannot be specified on the data line of an *INCIDENT WAVE INTERACTION option if the ACCELERATION AMPLITUDE parameter is used.

This parameter is valid only for transient planar incident waves using the *INCIDENT WAVE INTERACTION PROPERTY, TYPE=PLANE option. Reflected loads, using the *INCIDENT WAVE REFLECTION option, are not permitted in this case.

CONWEP

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to define an incident wave using the *CONWEP CHARGE PROPERTY option. This parameter is valid only for blast waves using the *INCIDENT WAVE INTERACTION PROPERTY, TYPE=AIR BLAST or TYPE=SURFACE BLAST option.

PRESSURE AMPLITUDE

Set this parameter equal to the name of the amplitude curve defining the fluid pressure time history at the standoff point ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual). The corresponding fluid traction, if required, will be computed from the pressure amplitude reference.

UNDEX

Include this parameter to define a spherical incident wave using the *UNDEX CHARGE PROPERTY option. This parameter is valid only for spherical incident waves using the *INCIDENT WAVE INTERACTION PROPERTY, TYPE=SPHERE option.

Optional, mutually exclusive parameters for matrix generation and steady-state dynamics analysis (direct or subspace):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading at the standoff point.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading at the standoff point.

Data line to define an incident wave when the CONWEP parameter is omitted:

First (and only) line:

1. Surface name.
2. The node number or set name of the source node.
3. The node number or set name of the standoff node.
4. Reference magnitude.

Data line to define an incident wave when the CONWEP parameter is included:

First (and only) line:

1. Surface name.
2. The node number or set name of the source node.
3. Blank field.
4. Time of detonation given in total time. See "Conventions," Section 1.2.2 of the Abaqus Analysis User's Manual, for a discussion of the total time measure.
5. Magnitude scale factor. The default is 1.0.

9.11 *INCIDENT WAVE INTERACTION PROPERTY: Define the geometric data and fluid properties describing an incident wave.

This option defines the geometric data and fluid properties used to define incident waves. Each *INCIDENT WAVE INTERACTION option must refer to an *INCIDENT WAVE INTERACTION PROPERTY definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *INCIDENT WAVE INTERACTION
- *UNDEX CHARGE PROPERTY
- *CONWEP CHARGE PROPERTY

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the incident wave interaction property in the *INCIDENT WAVE INTERACTION option.

Optional parameter:

TYPE

Set TYPE=PLANE (default) to specify a planar incident wave.

Set TYPE=SPHERE to specify a spherical incident wave.

Set TYPE=DIFFUSE to specify a field of planar waves incident from multiple angles.

Set TYPE=AIR BLAST to specify a spherical blast wave. This option is applicable only in an Abaqus/Explicit analysis.

Set TYPE=SURFACE BLAST to specify a hemispherical blast wave. This option is applicable only in an Abaqus/Explicit analysis.

Data line to define an incident wave interaction property with TYPE=PLANE, TYPE=SPHERE (acoustic decay), or TYPE=SPHERE used in conjunction with the *UNDEX CHARGE PROPERTY option:

First (and only) line:

1. c_f, the speed of sound in the fluid, defining the rate of propagation of the wave.
2. ρ_f, the fluid mass density.

Data line to define an incident wave interaction property with TYPE=SPHERE and generalized spatial decay:

First (and only) line:

1. c_f, the speed of sound in the fluid, defining the rate of propagation of the wave.
2. ρ_f, the fluid mass density.
3. A, dimensionless decay parameter ($A > -1$). The default is 0.
4. B, dimensionless decay parameter ($B > -1$). The default is 0.
5. C, dimensionless decay parameter ($C \geq 0$). The default is 0.

Data line to define incident wave loading from diffuse sources (TYPE=DIFFUSE):

First (and only) line:

1. c_f, the speed of sound in the fluid, defining the propagation speed of the waves.
2. ρ_f, the fluid mass density.
3. *N*, the seed number for the diffuse source calculation: N^2 sources will be used in the simulation.

No data lines are required for TYPE=AIR BLAST or TYPE=SURFACE BLAST used in conjunction with the *CONWEP CHARGE PROPERTY option.

9.12 *INCIDENT WAVE PROPERTY: Define the geometric data describing an incident wave.

The preferred interface for defining the geometric data for an incident wave is the *INCIDENT WAVE INTERACTION PROPERTY option used in conjunction with the *INCIDENT WAVE INTERACTION option. The alternative interface uses the *INCIDENT WAVE PROPERTY option to define the geometric data for incident waves.

Each *INCIDENT WAVE option must refer to an *INCIDENT WAVE PROPERTY definition. The *INCIDENT WAVE PROPERTY option must be followed by the *INCIDENT WAVE FLUID PROPERTY option, which defines the fluid properties used in the incident wave loading.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Unsupported; this option has been superseded by incident wave interaction properties.

References:

- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *INCIDENT WAVE
- *INCIDENT WAVE FLUID PROPERTY

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the incident wave property in the *INCIDENT WAVE option.

Optional parameter:

TYPE

Set TYPE=PLANE (default) to specify a planar incident wave.
Set TYPE=SPHERE to specify a spherical incident wave.

Data lines to define an incident wave property:

First line:

1. X-coordinate of $\mathbf{x}_0$, the position of the incident wave standoff point.
2. Y-coordinate of $\mathbf{x}_0$, the position of the incident wave standoff point.

3. *Z*-coordinate of $\mathbf{x_0}$, the position of the incident wave standoff point.
4. *X*-component of $\mathbf{v_0}$, the velocity of the incident wave standoff point.
5. *Y*-component of $\mathbf{v_0}$, the velocity of the incident wave standoff point.
6. *Z*-component of $\mathbf{v_0}$, the velocity of the incident wave standoff point.

Second line:

1. *X*-coordinate of $\mathbf{x_S}$, the position of the incident wave source point. Alternatively, specify the name of an *AMPLITUDE definition describing the time history of this coordinate.
2. *Y*-coordinate of $\mathbf{x_S}$, the position of the incident wave source point. Alternatively, specify the name of an *AMPLITUDE definition describing the time history of this coordinate.
3. *Z*-coordinate of $\mathbf{x_S}$, the position of the incident wave source point. Alternatively, specify the name of an *AMPLITUDE definition describing the time history of this coordinate.

If TYPE=PLANE, the vector from $\mathbf{x_S}$ to $\mathbf{x_0}$ defines the direction of the incoming wave; the distance between the two points is unimportant. For incident wave loads using bubble amplitudes, the source positions defined by the user with the *INCIDENT WAVE PROPERTY option are interpreted as the initial positions of the source.

9.13 *INCIDENT WAVE REFLECTION: Define the reflection load on a surface caused by incident wave fields.

This option is used to define reflected incident wave fields. It must be used in conjunction with the *INCIDENT WAVE INTERACTION option (preferred interface for applying incident wave loading) or the *INCIDENT WAVE option (alternative interface).

Products: Abaqus/Standard Abaqus/Explicit

Type: History data

Level: Step

References:

- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *INCIDENT WAVE INTERACTION
- *INCIDENT WAVE

There are no parameters associated with this option.

Data lines to define an incident wave reflection:

First line:

1. Distance from the source point to the first reflecting plane.
2. *X*-direction cosine of the normal to the reflecting plane, pointing away from the source point.
3. *Y*-direction cosine of the normal to the reflecting plane, pointing away from the source point.
4. *Z*-direction cosine of the normal to the reflecting plane, pointing away from the source point.
5. Reflection coefficient. The default value is zero, meaning no reflected wave will be applied.

Repeat this data line as often as necessary to describe the planes that reflect the incident wave.

9.14 *INCLUDE: Reference an external file containing Abaqus input data.

This option is used to reference an external file containing a portion of the Abaqus input file.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Part, Part instance, Assembly, Model, Step

Abaqus/CAE: Several input data options in Abaqus/CAE provide the capability to reference external files; for example, the material editor can read material properties from an ASCII file.

Reference:

- "Defining a model in Abaqus," Section 1.3.1 of the Abaqus Analysis User's Manual

Required parameter:

INPUT

Set this parameter equal to the name of the file containing the input data. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names.

Optional parameter:

PASSWORD

When the external file is encrypted, set this parameter equal to the file's password. Passwords are case-sensitive.

There are no data lines associated with this option.

9.15 *INCREMENTATION OUTPUT: Define output database requests for time incrementation data.

This option is used to write incrementation variables to the output database. It must be used in conjunction with the *OUTPUT, HISTORY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *OUTPUT

Optional parameter:

VARIABLE

Set VARIABLE=ALL to indicate that all incrementation variables applicable to this procedure type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default incrementation output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the incrementation variables requested for output must be specified on the data lines.

Data lines to request incrementation output:

First line:

1. Give the identifying keys for the variables to be written to the output database. The keys are defined in "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary to define the time incrementation variables to be written to the output database.

9.16 *INELASTIC HEAT FRACTION: Define the fraction of the rate of inelastic dissipation that appears as a heat source.

This option is used to provide for inelastic energy dissipation to act as a heat source in adiabatic thermo-mechanical problems. It is relevant when the ADIABATIC parameter is included on the *DYNAMIC or the *STATIC option. It is also relevant when a fully coupled thermal-stress analysis is conducted using the *COUPLED TEMPERATURE-DISPLACEMENT or *DYNAMIC TEMPERATURE-DISPLACEMENT options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Adiabatic analysis," Section 6.5.5 of the Abaqus Analysis User's Manual
- "Fully coupled thermal-stress analysis," Section 6.5.4 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data line to define the inelastic heat fraction:

First (and only) line:

1. Fraction of inelastic dissipation rate that appears as a heat flux per unit volume. The fraction may include a unit conversion factor if required. The default value is 0.9.

9.17 *INERTIA RELIEF: Apply inertia-based load balancing.

This option is used to apply inertia-based loads on a free or partially constrained body.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- "Inertia relief," Section 11.1.1 of the Abaqus Analysis User's Manual
- "Distributed loads," Section 30.4.3 of the Abaqus Analysis User's Manual

Optional parameters:

ORIENTATION

Set this parameter equal to the name given to the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) that specifies the orientation of the local system for rigid body degrees of freedom.

Optional, mutually exclusive parameters:

FIXED

Include this parameter (default) to indicate that the inertia relief load from a previous step should remain fixed at its value from the beginning of the current step.

REMOVE

Include this parameter to indicate that the inertia relief load from a previous step should be removed in the current step.

Optional data lines to specify global (or local, if the ORIENTATION parameter is used) degrees of freedom that define the free directions along which inertia relief loads are applied:

First line:

1. Integer list of degrees of freedom identifying the free directions.

Second line (only needed to define a reference point for the rigid body direction vectors when the user-chosen combination of free directions requires such a point):

1. Global *X*-coordinate of the reference point.

2. Global *Y*-coordinate of the reference point.
3. Global *Z*-coordinate of the reference point.

These data lines are needed only if rigid body motions are constrained in some directions.

There are no data lines when the FIXED or REMOVE parameters are specified.

9.18 *INITIAL CONDITIONS: Specify initial conditions for the model.

This option is used to prescribe initial conditions for an analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE Abaqus/Aqua

Type: Model data

Level: Model

Abaqus/CAE: Load module

Reference:

- "Initial conditions in Abaqus/Standard and Abaqus/Explicit," Section 30.2.1 of the Abaqus Analysis User's Manual

Required parameters:

TYPE

Set TYPE=ACOUSTIC STATIC PRESSURE to define initial static pressure values at acoustic nodes for use in evaluating the cavitation status of the acoustic element nodes in Abaqus/Explicit.

Set TYPE=CONCENTRATION to give initial normalized concentrations for a mass diffusion analysis in Abaqus/Standard.

Set TYPE=CONTACT to specify initial bonded contact conditions on part of the slave surface identified by a node set in an Abaqus/Standard analysis.

Set TYPE=ENRICHMENT to specify initial location of an enriched feature, such as a crack, in an Abaqus/Standard analysis. Two signed distance functions per node are generally required to describe the crack location, including the location of crack tips, in a cracked geometry. The first describes the crack surface while the second is used to construct an orthogonal surface so that the intersection of the two surfaces gives the crack front. The first signed distance function is assigned only to nodes of elements intersected by the crack while the second signed distance function is assigned only to nodes of elements containing the crack tips. No explicit representation of the crack is needed as the crack is entirely described by the nodal data.

Set TYPE=FIELD to specify initial values of field variables. The VARIABLE parameter can be used with this parameter to define the field variable number. The STEP and INC parameters can be used in conjunction with the FILE parameter to define initial values of field variables from a results or output database file. The STEP and INC parameters can also be used in conjunction with the FILE and OUTPUT VARIABLE parameters to define initial values of field variables based on scalar nodal output variables read from an output database file.

Set TYPE=FLUID PRESSURE to give initial pressures for hydrostatic fluid filled cavities.

Set TYPE=HARDENING to prescribe initial equivalent plastic strain and, if relevant, the initial backstress tensor or to prescribe initial volumetric compacting plastic strain for the crushable

foam model. The REBAR and, in Abaqus/Standard, SECTION POINTS and USER parameters can be used with this parameter. If the USER parameter is used, the initial conditions on equivalent plastic strain and, if relevant, the backstress tensor must be specified via user subroutine **HARDINI** for each section point and for each rebar. Consequently, in this case the REBAR and SECTION POINTS parameters do not have any effect and are ignored. If the USER parameter is omitted, Abaqus/Standard assumes that the initial conditions are defined on the data lines.

Set TYPE=INITIAL GAP to identify the elements within which tangential fluid flow exists initially.

Set TYPE=MASS FLOW RATE to specify initial values of mass flow rates in Abaqus/Standard heat transfer analyses involving forced convection modeled with the forced convection/diffusion heat transfer elements.

Set TYPE=PLASTIC STRAIN to specify initial plastic strains. The SECTION POINTS and REBAR parameters can be used with this parameter. It is assumed that the plastic strain components are defined on each data line in the order given for the element type, as defined in Part VI, "Elements," of the Abaqus Analysis User's Manual.

Set TYPE=PORE PRESSURE to give initial pore fluid pressures for a coupled pore fluid diffusion/stress analysis in Abaqus/Standard. The STEP and INC parameters can also be used with the FILE parameter to define initial values of pore fluid pressures based on scalar nodal output variables read from an output database file.

Set TYPE=POROSITY to give initial porosity values for materials defined with the *EOS COMPACTION option in Abaqus/Explicit.

Set TYPE=PRESSURE STRESS to give initial pressure stresses for a mass diffusion analysis in Abaqus/Standard. The STEP and INC parameters can be used in conjunction with the FILE parameter to define initial values of pressure stress from the results file of a previous Abaqus/Standard stress/displacement analysis.

Set TYPE=RATIO to give initial void ratio values for a coupled pore fluid diffusion/stress analysis in Abaqus/Standard. The USER parameter can be used with this parameter to define initial void ratio values in user subroutine **VOIDRI**.

Set TYPE=REF COORDINATE to define the reference mesh (initial metric) for membrane elements in Abaqus/Explicit using the element number and the coordinates of all of the nodes associated with the element. If a reference mesh is specified for an element, no initial stress or strain can be specified for the same element. The initial stress and strain are computed automatically to account for deformation from the reference to the initial configuration.

Set TYPE=NODE REF COORDINATE to define the reference mesh (initial metric) for membrane elements in Abaqus/Explicit using node numbers and the coordinates of each node. If a reference mesh is specified for an element, no initial stress or strain can be specified for the same element. The initial stress and strain are computed automatically to account for deformation from the reference to the initial configuration.

Set TYPE=RELATIVE DENSITY to give initial relative density values for materials defined with the *POROUS METAL PLASTICITY option.

Set TYPE=ROTATING VELOCITY to prescribe initial velocities in terms of an angular velocity and a global translational velocity.

Set TYPE=SATURATION to give initial saturation values for the analysis of flow through a porous medium in Abaqus/Standard. If no initial saturation values are given on this option, the default is fully saturated conditions (saturation of 1.0). For partial saturation the initial saturation and the pore fluid pressure must be consistent in the sense that the pore fluid pressure must lie within the range of absorption and exsorption values for the initial saturation value. If this is not the case, Abaqus/Standard will adjust the saturation value as needed to satisfy this requirement.

Set TYPE=SOLUTION to give initial values of solution-dependent state variables. The REBAR and, in Abaqus/Standard, USER parameters can be used with this parameter. If TYPE=SOLUTION is used without the USER parameter, element average quantities of the solution-dependent state variables must be defined on each data line.

Set TYPE=SPECIFIC ENERGY to give initial specific energy values for materials defined with the *EOS option in Abaqus/Explicit.

Set TYPE=SPUD EMBEDMENT to give the initial embedment for a spud can in an Abaqus/Aqua analysis.

Set TYPE=SPUD PRELOAD to give the initial preload value for a spud can in an Abaqus/Aqua analysis.

Set TYPE=STRESS to give initial stresses. (These stresses are effective stresses when the analysis includes pore fluid flow.) The GEOSTATIC; the REBAR; the SECTION POINTS; and, in Abaqus/Standard, the USER parameters can be used with this parameter. If TYPE=STRESS is used without the USER parameter, it is assumed that the stress components are defined on each data line in the order given for the element type, as defined in Part VI, “Elements,” of the Abaqus Analysis User’s Manual. The STEP and INC parameters can also be used with the FILE parameter to define initial stress values based on stress output variables read from an output database file.

Set TYPE=TEMPERATURE to give initial temperatures. The STEP and INC parameters can be used in conjunction with the FILE parameter to define initial temperatures from the results or output database file of a previous Abaqus/Standard heat transfer analysis.

Set TYPE=VELOCITY to prescribe initial velocities. Initial velocities should be defined in the global directions, regardless of the use of the *TRANSFORM option.

Set TYPE=VOLUME FRACTION to define the initial material content of Eulerian elements in an Abaqus/Explicit analysis.

STEP

This parameter is used only with the FILE parameter. The parameter specifies the step in the results file of a previous Abaqus analysis from which prescribed fields of TYPE=FIELD, TYPE=PRESSURE STRESS, or TYPE=TEMPERATURE are to be read. It can also specify the step in the output database file of a previous Abaqus analysis from which prescribed fields of TYPE=FIELD, TYPE=PORE PRESSURE, TYPE=STRESS, or TYPE=TEMPERATURE are to be read.

Optional parameters:

ABSOLUTE EXTERIOR TOLERANCE

This parameter is relevant only for use with the INTERPOLATE parameter. Set this parameter equal to the absolute value (given in the units used in the model) by which nodes of the current model may

lie outside the region of the model in the output database specified by the FILE parameter. If this parameter is not used or has a value of 0.0, the EXTERIOR TOLERANCE parameter will apply.

DEFINITION

Set DEFINITION=COORDINATES (default) to define the axis of rotation for TYPE=ROTATING VELOCITY by giving the coordinates of the two points, a and b.

Set DEFINITION=NODES to define the axis of rotation for TYPE=ROTATING VELOCITY by giving global node numbers for points a and b.

EXTERIOR TOLERANCE

This parameter is relevant only for use with the INTERPOLATE parameter. Set this parameter equal to the fraction of the average element size by which nodes of the current model may lie outside the region of the elements of the model in the output database specified by the FILE parameter. The default value is 0.05.

If both tolerance parameters are specified, Abaqus uses the tighter tolerance.

FILE

Set this parameter equal to the name of the results file or output database file from which initial field variable, stress, pore pressure or pressure stress data are to be read. This parameter must be used in conjunction with the STEP and INC parameters.

FULL TENSOR

Include this parameter if the kinematic shift tensor (backstress) components are specified using the full tensor format, regardless of the element type to which the initial conditions are applied.

This parameter can be used only in conjunction with the parameter TYPE=HARDENING. It cannot be used if any of the parameters REBAR, SECTION POINTS, or USER has been used.

GEOSTATIC

This parameter is used only with TYPE=STRESS to specify that a geostatic stress state, in which stresses vary with elevation only, is being defined.

INC

This parameter is used only with the FILE parameter. If the initial conditions are read from the results file, the INC parameter is required. If the initial conditions are read from the output database file, the INC parameter is optional; if omitted, the initial conditions will be read from the end of the step specified on the STEP parameter.

The parameter specifies the increment in the results file of a previous Abaqus analysis from which prescribed fields of TYPE=FIELD, TYPE=PRESSURE STRESS, or TYPE=TEMPERATURE are to be read. It can also specify the increment in the output database file of a previous Abaqus analysis from which prescribed fields of TYPE=FIELD, TYPE=PORE PRESSURE, TYPE=STRESS, or TYPE=TEMPERATURE are to be read.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

INTERPOLATE

Include this parameter in conjunction with the FILE, STEP, and INC parameters to indicate that the nodal temperatures being read into the temperature field or the scalar nodal output variable being read into a predefined field needs to be interpolated between dissimilar meshes. This feature is used to read nodal values from an output database file generated during a previous Abaqus analysis.

For temperature fields this parameter and the MIDSIDE parameter are mutually exclusive. For temperature fields if the initial analysis uses first-order elements and the current mesh is the same but uses second-order elements, use the MIDSIDE parameter instead. The MIDSIDE parameter is not supported for predefined fields; therefore, the INTERPOLATE parameter is the only option for initializing predefined fields using scalar nodal output values from a dissimilar mesh.

MIDSIDE

This parameter applies only to Abaqus/Standard analyses.

Include this parameter in conjunction with the FILE, STEP, and INC parameters to indicate that midside node temperatures in second-order elements are to be interpolated from corner node temperatures. This feature is used to read temperatures from a results or output database file generated during a heat transfer analysis using first-order elements. This parameter and the INTERPOLATE parameter are mutually exclusive.

NORMAL

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only with TYPE=CONTACT to specify that the nodes in the node set (or the contact pair, if a node set is not defined) are bonded only in the normal (contact) direction and are allowed to move freely in the tangential direction. If the nodes in the node set (or the contact pair) are to be bonded in all directions, this parameter should be omitted.

NUMBER BACKSTRESSES

Set this parameter equal to the number of backstresses. This parameter can be used only in conjunction with TYPE=HARDENING. The default number of backstresses is 1, and the maximum allowed is 10.

OUTPUT VARIABLE

This parameter is required when TYPE=FIELD and the FILE parameter references an output database.

Set this parameter equal to the scalar nodal output variable that will be read from an output database and used to initialize a specified predefined field. For a list of scalar nodal output variables that can be used to initialize a predefined field, see "Predefined fields," Section 30.6.1 of the Abaqus Analysis User's Manual.

REBAR

This parameter can be used with TYPE=HARDENING, TYPE=SOLUTION, or TYPE=STRESS.

When used with TYPE=HARDENING, it specifies that rebars are in a work hardened state, with initial equivalent plastic strain and, possibly, initial backstress.

When used with TYPE=PLASTIC STRAIN, it specifies the initial plastic strain in the rebar.

When used with TYPE=SOLUTION, it specifies that rebars are being assigned initial solution-dependent state variable values.

When used with TYPE=STRESS, it specifies that prestress in rebars is being defined. When performing an Abaqus/Standard analysis, some iteration will usually be needed in this case to establish a self-equilibrating stress state in the rebar and concrete. The *PRESTRESS HOLD option can be useful for post-tensioning simulations (see "Defining rebar as an element property," Section 2.2.4 of the Abaqus Analysis User's Manual).

SECTION POINTS

This parameter is used only with TYPE=PLASTIC STRAIN, TYPE=STRESS, and TYPE=HARDENING to specify plastic strains, total mechanical strains, stresses, and hardening variables at individual section points through the thickness of a shell element. This parameter can be used only when shell properties are defined using the *SHELL SECTION option. It cannot be used when properties are defined using the *SHELL GENERAL SECTION option.

UNBALANCED STRESS

This parameter applies only to Abaqus/Standard analyses.

This parameter is used only with TYPE=STRESS.

Set UNBALANCED STRESS=RAMP (default) if the unbalanced stress is to be resolved linearly over the step.

Set UNBALANCED STRESS=STEP if the unbalanced stress is to be resolved in the first increment.

USER

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used with TYPE=HARDENING, TYPE=PORE PRESSURE, TYPE=RATIO, TYPE=SOLUTION, or TYPE=STRESS.

When used with TYPE=HARDENING, it specifies that the initial conditions on equivalent plastic strain and, if relevant, backstress tensor are to be given via user subroutine **HARDINI**.

When used with TYPE=PORE PRESSURE, it specifies that initial pore pressures are to be given via user subroutine **UPOREP**.

When used with TYPE=RATIO, it specifies that initial void ratios are to be given via user subroutine **VOIDRI**.

When used with TYPE=SOLUTION, it specifies that initial solution-dependent state variable fields are to be given via user subroutine **SDVINI**.

When used with TYPE=STRESS, it specifies that stresses are to be given via user subroutine **SIGINI**.

VARIABLE

This parameter is used only with TYPE=FIELD when it is used to define the field variable number. The default is VARIABLE=1. Any number of separate field variables can be used: each must be numbered consecutively (1, 2, 3, etc.)

Data line for TYPE=ACOUSTIC STATIC PRESSURE:

First (and only) line:

1. Node set or node number.
2. Hydrostatic pressure at the first reference point.
3. *X*-coordinate of the first reference point.
4. *Y*-coordinate of the first reference point.
5. *Z*-coordinate of the first reference point.
6. Hydrostatic pressure at the second reference point.
7. *X*-coordinate of the second reference point.
8. *Y*-coordinate of the second reference point.
9. *Z*-coordinate of the second reference point.

Data lines for TYPE=CONCENTRATION:

First line:

1. Node set or node number.
2. Initial normalized concentration value at the node.

Repeat this data line as often as necessary to define the initial normalized concentration at various nodes or node sets.

Data lines for TYPE=CONTACT:

First line:

1. Slave surface name.
2. Master surface name.
3. Name of the node set associated with the slave surface.

Repeat this data line as often as necessary to define partially bonded surfaces.

Data lines for TYPE=ENRICHMENT:

First line:

1. Element number or element set label.
2. Relative position of the node forming the element connectivity.
3. Name of the enriched feature specified on the *ENRICHMENT option.
4. Value of first signed distance function.

5. Value of second signed distance function. Leave this entry blank if only the first signed distance function is needed.

Repeat this data line as often as necessary to define initial signed distance functions in various elements or element sets. The signed distance functions must be defined at all nodes within an element.

Data lines for TYPE=FIELD, VARIABLE=*n*:

First line:

1. Node set or node number.
2. Initial value of this field variable at the first temperature point. For shells and beams several values (or a value and the field variable gradients across the section) can be given at each node (see "Beam modeling: overview," Section 26.3.1 of the Abaqus Analysis User's Manual, as well as "Shell elements: overview," Section 26.6.1 of the Abaqus Analysis User's Manual). For heat transfer shells the field variables at each temperature point through the shell thickness must be specified. The number of values depends on the (maximum) number of points specified on the data lines associated with the *SHELL SECTION options.
3. Initial value of this field variable at the second temperature point.
4. Etc., up to seven values.

Subsequent lines (only needed if initial values must be specified at more than seven temperature points at any node):

1. Eighth initial value of this field variable at this temperature point.
2. Etc., up to eight initial values per line.

It may be necessary to leave blank data lines for some nodes if any other node in the model has more than seven field variable points because the total number of field variables that Abaqus expects to read for any node is based on the maximum number of field variable values for all the nodes in the model. These trailing initial values will be zero and will not be used in the analysis.

Repeat this set of data lines as often as necessary to define initial temperatures at various nodes or node sets.

No data lines are required for TYPE=FIELD, VARIABLE=*n*, FILE=*file*, STEP=*step*, INC=*inc*.

Data lines for TYPE=FLUID PRESSURE:

First line:

1. Node set or node number of fluid cavity reference node.
2. Fluid pressure.

Repeat this data line as often as necessary to define initial fluid pressure for various fluid-filled cavities.

Data lines to prescribe initial equivalent plastic strain or backstresses using TYPE=HARDENING if the REBAR, SECTION POINTS, and USER parameters are omitted:

First line:

1. Element number or element set label.
2. Initial equivalent plastic strain, $\bar{\varepsilon}^{pl}|_0$.
3. First value of the initial first backstress, $\alpha 1^0_{11}$.
4. Second value of the initial first backstress, $\alpha 1^0_{22}$.
5. Etc., up to six backstress components.

Subsequent lines (only needed if the NUMBER BACKSTRESSES parameter has a value greater than one):

1. First value of the initial second backstress, $\alpha 2^0_{11}$.
2. Second value of the initial second backstress, $\alpha 2^0_{22}$.
3. Etc., backstress components for each backstress must be specified on a separate data line.

The backstress components are relevant only for the kinematic hardening models. Give the backstress components as defined for this element type in Part VI, "Elements," of the Abaqus Analysis User's Manual. Values given on the data lines are applied uniformly over the element. In any element for which an *ORIENTATION option applies, backstresses must be given in the local system ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual).

Repeat this set of data lines as often as necessary to define the hardening parameters for various elements or element sets.

Data lines to prescribe initial volumetric compacting plastic strain for the crushable foam model using TYPE=HARDENING:

First line:

1. Element number or element set label.
2. Initial volumetric compacting plastic strain, $-\varepsilon^{pl}_{\text{vol}}$.

Repeat this data line as often as necessary to define the initial volumetric compacting plastic strain for various elements or element sets.

Data lines for TYPE=HARDENING, REBAR:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the initial conditions will be applied to all rebars in the model.
3. Initial equivalent plastic strain, $\bar{\varepsilon}^{pl}|_0$.
4. Initial first backstress, $\alpha 1^0_{11}$. (Only relevant for the kinematic hardening models.)

Subsequent lines (only needed if the NUMBER BACKSTRESSES parameter has a value greater than one):

1. Initial second backstress, $\alpha 2^0_{11}$. (Only relevant for the kinematic hardening models.)
2. Etc., backstress components for each backstress must be specified on a separate data line.

Repeat this set of data lines as often as necessary to define the hardening parameters for rebars in various elements or element sets.

No data lines are required for TYPE=HARDENING, USER.

Data lines for TYPE=HARDENING, SECTION POINTS:

First line:

1. Element number or element set label.
2. Section point number.
3. Initial equivalent plastic strain, $\bar{\varepsilon}^{pl}|_0$.
4. First value of the first initial backstress, $\alpha 1^0_{11}$. (Only relevant for the kinematic hardening models.)
5. Second value of the first initial backstress, $\alpha 1^0_{22}$.
6. Third value of the first initial backstress, $\alpha 1^0_{12}$.

Subsequent lines (only needed if the NUMBER BACKSTRESSES parameter has a value greater than one):

1. First value of the initial second backstress, $\alpha 2^0_{11}$.
2. Second value of the initial second backstress, $\alpha 2^0_{22}$.
3. Etc., backstress components for each backstress must be specified on a separate data line.

The backstress components are relevant only for the kinematic hardening model. Give the backstress components as defined for this element type in Part VI, "Elements," of the Abaqus Analysis User's Manual. In any element for which an *ORIENTATION option applies, the backstress components must be given in the local system ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual).

Repeat this set of data lines as often as necessary to define the hardening parameters in various elements or element sets. The hardening parameters must be defined at all section points within an element.

Data lines for TYPE=INITIAL GAP:

First line:

1. Element number or element set label.

Repeat this data line as often as necessary to identify various elements or element sets.

Data lines for TYPE=MASS FLOW RATE:

First line:

1. Node set or node number.
2. Initial mass flow rate per unit area in the x-direction or total initial mass flow rate in the cross-section for one-dimensional elements.
3. Initial mass flow rate per unit area in the y-direction (not needed for nodes associated with one-dimensional convective flow elements).
4. Initial mass flow rate per unit area in the z-direction (not needed for nodes associated with one-dimensional convective flow elements).

Repeat this data line as often as necessary to define mass flow rates at various nodes or node sets.

Data lines for TYPE=NODE REF COORDINATE:

First line:

1. Node number.
2. X-coordinate of the node.
3. Y-coordinate of the node.
4. Z-coordinate of the node.

Repeat this data line as often as necessary to define the initial coordinates of the mesh using nodal coordinates.

Data lines to prescribe initial plastic strains using TYPE=PLASTIC STRAIN if the REBAR and SECTION POINTS parameters are omitted:

First line:

1. Element number or element set label.
2. Value of first plastic strain component, ε_{11}^{pl}.
3. Value of second plastic strain component, ε_{22}^{pl}.
4. Etc., up to six plastic strain components.

Give the plastic strain components as defined for this element type in Part VI, "Elements," of the Abaqus Analysis User's Manual. Values given on the data lines are applied uniformly over the element. In any element for which an *ORIENTATION option applies, the plastic strains must be given in the local system ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual).

Repeat this data line as often as necessary to define initial plastic strains in various elements or element sets.

Data lines for TYPE=PLASTIC STRAIN, REBAR:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the initial conditions will be applied to all rebars in the model.
3. Initial plastic strain value.

Repeat this data line as often as necessary to define the initial plastic strain in the rebars of various elements or element sets.

Data lines for TYPE=PLASTIC STRAIN, SECTION POINTS:

First line:

1. Element number or element set label.
2. Section point number.
3. Value of first plastic strain component, ε_{11}^{pl}.
4. Value of second plastic strain component, ε_{22}^{pl}.
5. Value of third plastic strain component, ε_{12}^{pl}.

Give the initial plastic strain components as defined for this element type in Part VI, "Elements," of the Abaqus Analysis User's Manual. In any element for which an *ORIENTATION option applies, the plastic strain components must be given in the local system ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual).

Repeat this data line as often as necessary to define initial plastic strains in various elements or element sets. Plastic strains must be defined at all section points within an element.

Data lines for TYPE=PORE PRESSURE if the USER parameter is omitted:

First line:

1. Node set or node number.
2. First value of fluid pore pressure, u_w.
3. Vertical coordinate corresponding to the above value.
4. Second value of fluid pore pressure, u_w.
5. Vertical coordinate corresponding to the above value.

Omit the elevation values and the second pore pressure value to define a constant pore pressure distribution.

Repeat this data line as often as necessary to define the fluid pore pressure at various nodes or node sets.

No data lines are required for TYPE=PORE PRESSURE, USER.

No data lines are required for TYPE=PORE PRESSURE, FILE=*file*, STEP=*step*, INC=*inc*.

Data lines for TYPE=POROSITY:

First line:

1. Element number or element set label.
2. Initial porosity.

Repeat this data line as often as necessary to define initial porosity in various elements or element sets.

Data lines for TYPE=PRESSURE STRESS:

First line:

1. Node set or node number.
2. Equivalent pressure stress, p.

Repeat this data line as often as necessary to define the pressure stress at various nodes or node sets.

No data lines are required for TYPE=PRESSURE STRESS, FILE=*file*, STEP=*step*, INC=*inc*.

Data lines for TYPE=RATIO if the USER parameter is omitted:

First line:

1. Node set or node number.
2. First value of void ratio.
3. Vertical coordinate corresponding to the above value.
4. Second value of void ratio.
5. Vertical coordinate corresponding to the above value.

Omit the elevation values and the second void ratio value to define a constant void ratio distribution.

Repeat this data line as often as necessary to define void ratios at various nodes or node sets.

No data lines are required for TYPE=RATIO, USER.

Data lines for TYPE=REF COORDINATE:

First line:

1. Element number.
2. X-coordinate of the first node.
3. Y-coordinate of the first node.
4. Z-coordinate of the first node.
5. X-coordinate of the second node.

6. *Y*-coordinate of the second node.
7. *Z*-coordinate of the second node.

Second line:

1. *X*-coordinate of the third node.
2. *Y*-coordinate of the third node.
3. *Z*-coordinate of the third node.
4. *X*-coordinate of the fourth node.
5. *Y*-coordinate of the fourth node.
6. *Z*-coordinate of the fourth node.

Repeat this pair of data lines as often as necessary to define the reference mesh in various elements. The order of the nodal coordinates must be consistent with the element connectivity.

Data lines for TYPE=RELATIVE DENSITY:

First line:

1. Node set or node number.
2. Initial relative density.

Repeat this data line as often as necessary to define initial relative density at various nodes or node sets.

Data lines for TYPE=ROTATING VELOCITY, DEFINITION=COORDINATES:

First line:

1. Node set or node number.
2. Angular velocity about the axis defined from point *a* to point *b*, where the coordinates of *a* and *b* are given below.
3. Global *X*-component of translational velocity.
4. Global *Y*-component of translational velocity.
5. Global *Z*-component of translational velocity.

Second line:

1. Global *X*-component of point *a* on the axis of rotation.
2. Global *Y*-component of point *a* on the axis of rotation.
3. Global *Z*-component of point *a* on the axis of rotation.
4. Global *X*-component of point *b* on the axis of rotation.
5. Global *Y*-component of point *b* on the axis of rotation.
6. Global *Z*-component of point *b* on the axis of rotation.

Repeat this pair of data lines as often as necessary to define the angular and translational velocities at various nodes or node sets.

Data lines for TYPE=ROTATING VELOCITY, DEFINITION=NODES:

First line:

1. Node set or node number.
2. Angular velocity about the axis defined from point a to point b, where the coordinates of a and b are given below.
3. Global X-component of translational velocity.
4. Global Y-component of translational velocity.
5. Global Z-component of translational velocity.

Second line:

1. Node number of the node at point a.
2. Node number of the node at point b.

Repeat this pair of data lines as often as necessary to define the angular and translational velocities at various nodes or node sets.

Data lines for TYPE=SATURATION:

First line:

1. Node set or node number.
2. Saturation value, s. Default is 1.0.

Repeat this data line as often as necessary to define saturation at various nodes or node sets.

Data lines for TYPE=SOLUTION if the USER and REBAR parameters are omitted:

First line:

1. Element number or element set label.
2. Value of first solution-dependent state variable.
3. Value of second solution-dependent state variable.
4. Etc., up to seven solution-dependent state variables.

Subsequent lines (only needed if more than seven solution-dependent state variables exist in the model):

1. Value of eighth solution-dependent state variable.
2. Etc., up to eight solution-dependent state variables per line.

It may be necessary to leave blank data lines for some elements if any other element in the model has more solution-dependent state variables because the total number of variables that Abaqus expects to read for any element is based on the maximum number of solution-dependent state variables for all the elements in the model. These trailing initial values will be zero and will not be used in the analysis. Values given on the data lines will be applied uniformly over the element.

Repeat this set of data lines as often as necessary to define initial values of solution-dependent state variables for various elements or element sets.

Data lines for TYPE=SOLUTION, REBAR:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the solution-dependent state variables are applied to all rebars in these elements.
3. Value of first solution-dependent state variable.
4. Value of second solution-dependent state variable.
5. Etc., up to six solution-dependent state variables.

Subsequent lines (only needed if more than six solution-dependent state variables exist in the model):

1. Value of seventh solution-dependent state variable.
2. Etc., up to eight solution-dependent state variables per line.

It may be necessary to leave blank data lines for some elements if any other element in the model has more solution-dependent state variables because the total number of variables that Abaqus expects to read for any element is based on the maximum number of solution-dependent state variables for all the elements in the model. These trailing initial values will be zero and will not be used in the analysis. Values given on the data lines will be applied uniformly over the element.

Repeat this set of data lines as often as necessary to define initial values of solution-dependent state variables for various elements or element sets.

No data lines are required for TYPE=SOLUTION, USER.

Data lines for TYPE=SPECIFIC ENERGY:

First line:

1. Element number or element set label.
2. Initial specific energy.

Repeat this data line as often as necessary to define initial specific energy in various elements or element sets.

Data lines for TYPE=SPUD EMBEDMENT:

First line:

1. Element set or element number.
2. Spud can embedment, ν_i.

Repeat this data line as often as necessary to define initial embedment for various elements or element sets.

Data lines for TYPE=SPUD PRELOAD:

First line:

1. Element set or element number.
2. Spud can preload, $V_c^{(i)}$.

Repeat this data line as often as necessary to define initial preload for various elements or element sets.

Data lines for TYPE=STRESS if the GEOSTATIC, REBAR, SECTION POINTS, and USER parameters are omitted:

First line:

1. Element number or element set label.
2. Value of first (effective) stress component, axial force when used with the *BEAM GENERAL SECTION or *FRAME SECTION options, or direct membrane force per unit width in the local 1-direction when used with the *SHELL GENERAL SECTION option.
3. Value of second stress component.
4. Etc., up to six stress components.

Give the stress components as defined for this element type in Part VI, "Elements," of the Abaqus Analysis User's Manual. Stress values given on data lines are applied uniformly and equally over all integration points of the element. In any element for which an *ORIENTATION option applies, the stresses must be given in the local system ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual).

Repeat this data line as often as necessary to define initial stresses in various elements or element sets.

Data lines for TYPE=STRESS, GEOSTATIC:

First line:

1. Element number or element set label.
2. First value of vertical component of (effective) stress.
3. Vertical coordinate corresponding to the above value.
4. Second value of vertical component of (effective) stress.
5. Vertical coordinate corresponding to the above value.
6. First coefficient of lateral stress. This coefficient defines the x-direction stress components.
7. Second coefficient of lateral stress. This coefficient defines the y-direction stress component in three-dimensional cases and the thickness or hoop direction component in plane or axisymmetric cases. If this value is omitted, it is assumed to be the same as the first lateral stress coefficient given in the previous field.

Repeat this data line as often as necessary to define an initial geostatic stress state in various elements or element sets.

Data lines for TYPE=STRESS, REBAR:

First line:

1. Element number or element set label.
2. Rebar name. If this field is left blank, the stress is applied to all rebars in these elements.
3. Prestress value.

Repeat this data line as often as necessary to define initial stress in the rebars of various elements or element sets.

Data lines for TYPE=STRESS, SECTION POINTS:

First line:

1. Element number or element set label.
2. Section point number.
3. Value of first stress component.
4. Value of second stress component.
5. Etc., up to three stress components.

Give the stress components as defined for this element type in Part VI, "Elements," of the Abaqus Analysis User's Manual. Stress values given on data lines are applied uniformly over the element. In any element for which an *ORIENTATION option applies, the stresses must be given in the local system ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual).

Repeat this data line as often as necessary to define initial stresses in various elements or element sets. Stresses must be defined at all section points within an element.

No data lines are required for TYPE=STRESS, USER.
No data lines are required for TYPE=STRESS, FILE=*file*, STEP=*step*, INC=*inc*.

Data lines for TYPE=TEMPERATURE:

First line:

1. Node set or node number.
2. First initial temperature value at the node or node set. For shells and beams several values (or a value and the temperature gradients across the section) can be given at each node (see "Using a beam section integrated during the analysis to define the section behavior," Section 26.3.6 of the Abaqus Analysis User's Manual; "Using a general beam section to define the section behavior," Section 26.3.7 of the Abaqus Analysis User's Manual; "Using a shell section integrated during the analysis to define the section behavior," Section 26.6.5 of the Abaqus Analysis User's Manual; and "Using a general shell section to define the section behavior," Section 26.6.6 of the Abaqus Analysis User's Manual). For heat transfer shells the temperature at each point through the shell thickness must be specified. The number of values

depends on the (maximum) number of points specified on the data lines associated with the *SHELL SECTION options.

3. Second initial temperature value at the node or node set.
4. Etc., up to seven initial temperature values at this node or node set.

Subsequent lines (only needed if there are more than seven temperature values at any node):

1. Eighth initial temperature value at this node or node set.
2. Etc., up to eight initial temperature values per line.

If more than seven temperature values are needed at any node, continue on the next line. It may be necessary to leave blank data lines for some nodes if any other node in the model has more than seven temperature points because the total number of temperatures that Abaqus expects to read for any node is based on the maximum number of temperature values of all the nodes in the model. These trailing initial values will be zero and will not be used in the analysis.

Repeat this data line (or set of lines) as often as necessary to define initial temperatures at various nodes or node sets.

No data lines are required for TYPE=TEMPERATURE, FILE=*file*, STEP=*step*, INC=*inc*.

Data lines for TYPE=VELOCITY:

First line:

1. Node set or node number.
2. Degree of freedom.
3. Value of initial velocity.

Repeat this data line as often as necessary to define the initial velocity at various nodes or node sets.

Data lines for TYPE=VOLUME FRACTION:

First line:

1. Eulerian element number or element set label.
2. Name of the material instance as defined in the *EULERIAN SECTION.
3. Initial volume fraction, EVF, for this material ($0.0 < \text{EVF} \leq 1.0$). EVF=0.0 indicates that none of this material is present in the element, while EVF=1.0 indicates that the element is completely full of this material.

Repeat this data line as often as necessary to define the initial geometry of all Eulerian material instances. An element may appear in more than one data line if it initially contains more than one material. Elements are filled incrementally by reading the data lines in the input file from bottom to top; once the volume fraction for an element reaches one, additional volume fractions assigned to that element are ignored. If the final volume fraction for an element is less than one, the remainder of that element is filled with void; similarly, uninitialized elements are filled with void.

9.19 *INSTANCE: Begin an instance definition.

This option is used to instance a part within an assembly. It must be used in conjunction with the *ASSEMBLY and *END INSTANCE options. If the instance is not imported from a previous analysis, the *INSTANCE option must be used in conjunction with the *PART option. When importing a part instance from a previous analysis, the *INSTANCE option must be used in conjunction with the *IMPORT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Assembly

Abaqus/CAE: Assembly module for part instances not imported from a previous analysis; Load module for part instances imported from a previous analysis

References:

- "Defining an assembly," Section 2.9.1 of the Abaqus Analysis User's Manual
- "Transferring results between Abaqus analyses: overview," Section 9.2.1 of the Abaqus Analysis User's Manual
- *END INSTANCE
- *IMPORT

Required parameters if the instance is not imported from a previous analysis:

NAME

Set this parameter equal to a label that will be used to refer to the instance.

PART

Set this parameter equal to the name of the part being instanced.

Required parameter if the instance is to be imported from a previous analysis:

INSTANCE

Set this parameter equal to the name of the instance to be imported from the previous analysis.

Optional import parameter:

LIBRARY

Set this parameter equal to the name of the previous analysis from which the instance should be imported. The previous analysis output database (`.odb`) file should reside in the current (working) directory. If the LIBRARY parameter is omitted, the job name of the previous analysis must be specified on the command line using the **oldjob** parameter (see "Abaqus/Standard, Abaqus/Explicit,

and Abaqus/CFD execution," Section 3.2.2 of the Abaqus Analysis User's Manual). If both methods are used, the command line specification will take precedence over the LIBRARY parameter.

Data line to translate an instance that is not imported from a previous analysis:

First (and only) line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Data lines to translate and/or rotate an instance that is not imported from a previous analysis:

First line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Enter values of zero to apply a pure rotation.

Second line:

1. *X*-coordinate of point *a* on the axis of rotation (see Figure 9.19–1).
2. *Y*-coordinate of point *a* on the axis of rotation.
3. *Z*-coordinate of point *a* on the axis of rotation.
4. *X*-coordinate of point *b* on the axis of rotation.
5. *Y*-coordinate of point *b* on the axis of rotation.
6. *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

If both translation and rotation are specified, translation is applied before rotation.

There are no data lines for an instance that is imported from a previous analysis.

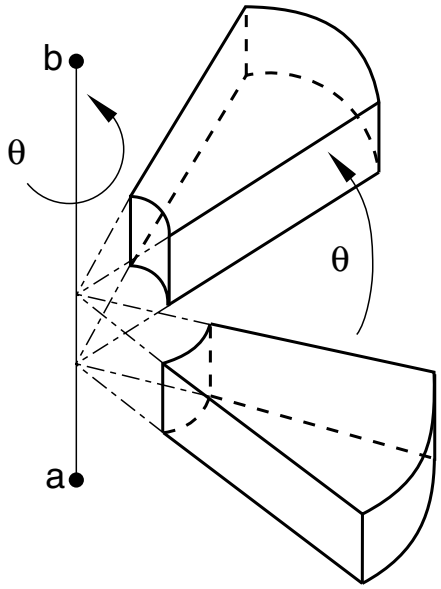


Figure 9.19–1 Rotation of an instance.

9.20 *INTEGRATED OUTPUT: Specify variables integrated over a surface to be written to the output database.

This option is used to write integrated quantities over a surface, such as the total force transmitted across a surface, to the output database. It must be used in conjunction with the *OUTPUT, HISTORY option.

Products: Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- "Integrated output section definition," Section 2.5.1 of the Abaqus Analysis User's Manual
- *OUTPUT
- *SURFACE
- *INTEGRATED OUTPUT SECTION

Required, mutually exclusive parameters:

SECTION

Set this parameter equal to the name of the *INTEGRATED OUTPUT SECTION (see "Integrated output section definition," Section 2.5.1 of the Abaqus Analysis User's Manual) over which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface (see "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Manual) over which this output request is being made.

ELSET

Set this parameter equal to the name of the element set over which this output request is being made.

Optional parameter:

VARIABLE

Set VARIABLE=ALL to indicate that all integrated output variables applicable to this procedure should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default integrated output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the integrated output variables for output must be specified on the data line.

Data lines to request integrated output:

First line:

1. Specify the identifying keys for the output variables to be written to the output database. The keys are defined in "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary to define the list of variables to be written to the output database.

9.21 *INTEGRATED OUTPUT SECTION: Define an integrated output section over a surface with a local coordinate system and a reference point.

This option is used to associate a surface with a coordinate system and/or a reference node to track the average motion of the surface. It can also be used in conjunction with an integrated output request to obtain output of quantities integrated over a surface.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Step module

References:

- "Integrated output section definition," Section 2.5.1 of the Abaqus Analysis User's Manual
- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *INTEGRATED OUTPUT
- *SURFACE

Required parameters:

NAME

Set this parameter equal to a label that will be used to refer to the integrated output section.

SURFACE

Set this parameter equal to the name of the surface (see "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Manual) to be associated with the integrated output section.

Optional parameters:

ORIENTATION

Set this parameter equal to the name of an orientation definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) to define the initial coordinate system for the section. This initial system can be further modified by using the PROJECT ORIENTATION parameter.

If this parameter is omitted, the global coordinate system is used.

POSITION

This parameter is relevant only if the REF NODE parameter is included.

Set POSITION=INPUT (default) if the location of the reference node is to be defined by the user.

Set POSITION=CENTER if the reference node is to be relocated from the user-defined location to the center of the surface in the initial configuration.

PROJECT ORIENTATION

Set PROJECT ORIENTATION=NO (default) if the initial coordinate system of the section should not be projected onto the section surface. If the ORIENTATION parameter is included, this choice results in an initial coordinate system that matches the defined orientation. If an orientation is not specified, the initial coordinate system matches the global coordinate system.

Set PROJECT ORIENTATION=YES if the initial coordinate system of the section should be modified by projecting onto the section surface using the average normal to the surface. If an orientation is not specified, the global coordinate system is projected onto the section surface.

REF NODE

Set this parameter equal to either the node number of the integrated output section reference node or to the name of a node set containing the reference node. If the name of a node set is chosen, the node set must contain exactly one node.

REF NODE MOTION

This parameter is relevant only if the REF NODE parameter is included.

Set REF NODE MOTION=INDEPENDENT (default) if the motion of the reference node is not based on the average motion of the surface.

Set REF NODE MOTION=AVERAGE TRANSLATION if the reference node must translate with the average translation of the surface. This choice is relevant only if the reference node is not connected to the rest of the model.

Set REF NODE MOTION=AVERAGE if the reference node must both translate and rotate with the average motion of the surface. This choice is relevant only if the reference node is not connected to the rest of the model.

There are no data lines associated with this option.

9.22 *INTERFACE: Define properties for contact elements.

This option is used to assign element section properties to ITT-, ISL-, IRS-, and ASI-type contact elements.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Property module; supported for two-dimensional, three-dimensional, and axisymmetric acoustic interface elements. Contact elements are not supported.

References:

- "Acoustic interface elements," Section 29.14.1 of the Abaqus Analysis User's Manual
- "Tube-to-tube contact elements," Section 36.3.1 of the Abaqus Analysis User's Manual
- "Slide line contact elements," Section 36.4.1 of the Abaqus Analysis User's Manual
- "Rigid surface contact elements," Section 36.5.1 of the Abaqus Analysis User's Manual

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the ITT-, ISL-, IRS-, and ASI-type contact elements for which properties are being defined.

Optional parameter:

NAME

Set this parameter equal to a label that will be used to refer to this interface definition. The label given can be used to identify this particular interface definition in user subroutines such as **GAPCON**.

Data line for ITT-type elements:

First (and only) line:

1. Radial clearance between the pipes.

Data lines for ISL21A and ISL22A elements:

There are no data lines.

Data lines for IRS-type elements for use with axisymmetric elements:

There are no data lines.

Data line for ASI1 elements:

First (and only) line:

1. Area associated with the elements.

Enter the direction cosine, in terms of the global Cartesian coordinate system, of the interface normal that points into the acoustic fluid:

2. *X*-direction cosine.
3. *Y*-direction cosine.
4. *Z*-direction cosine.

Data line for ASI-type elements for use with 2-D elements:

First (and only) line:

1. Element thickness. The default is unit thickness.

Data lines for ASI-type elements for use with axisymmetric elements or 3-D elements:

There are no data lines.

9.23 *ITS: Define properties for ITS elements.

This option is used to define the properties for ITS-type elements. The *DASHPOT, *FRICTION, and *SPRING options must immediately follow this option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Unsupported; similar functionality (with the exception of friction) is available by modeling connectors.

References:

- "Rigid surface contact elements," Section 36.5.1 of the Abaqus Analysis User's Manual
- *DASHPOT
- *FRICTION
- *SPRING

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the ITS-type elements for which properties are being defined.

Data line for ITSUNI elements:

First (and only) line:

1. Tube outside diameter.
2. Distance between the parallel support plates on opposite sides of the tube.
3. *X*-direction cosine of the axis of the tube.
4. *Y*-direction cosine of the axis of the tube.
5. *Z*-direction cosine of the axis of the tube.
6. *X*-direction cosine of the normal to either one of the support plates.
7. *Y*-direction cosine of the normal to either one of the support plates.
8. *Z*-direction cosine of the normal to either one of the support plates.

Data line for ITSCYL elements:

First (and only) line:

1. Tube outside diameter.
2. Diameter of the hole in the support plate.
3. *X*-direction cosine of the axis of the tube.
4. *Y*-direction cosine of the axis of the tube.
5. *Z*-direction cosine of the axis of the tube.

10. J

10.1 *JOINT: Define properties for JOINTC elements.

This option is used to define the properties for JOINTC elements. The *DASHPOT and *SPRING options must immediately follow this option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Unsupported; similar functionality is available by modeling connectors.

References:

- "Flexible joint element," Section 29.3.1 of the Abaqus Analysis User's Manual
- *DASHPOT
- *SPRING

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the JOINTC elements for which properties are being defined.

Optional parameter:

ORIENTATION

Set this parameter equal to the name given to the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) that specifies the initial orientation of the local system in the joint.

There are no data lines associated with this option; instead, include *SPRING and *DASHPOT options as needed to define the joint behavior.

10.2 *JOINT ELASTICITY: Specify elastic properties for elastic-plastic joint elements.

This option is used to define linear elastic moduli for elastic-plastic joint elements. It can be used only in conjunction with the *EPJOINT option.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Elastic-plastic joints," Section 29.11.1 of the Abaqus Analysis User's Manual
- *EPJOINT

Required parameters:

MODULI

Set MODULI=SPUD CAN to define spud can moduli. Set MODULI=GENERAL to enter a general elastic modulus.

NDIM

Set NDIM=2 to enter values for a two-dimensional problem. Set NDIM=3 to enter values for a three-dimensional problem.

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the moduli. If this parameter is omitted, it is assumed that the moduli are constant or depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines for MODULI=SPUD CAN and NDIM=2:

First line:

1. G_{vv}, equivalent elastic shear modulus for vertical displacements.
2. G_{hh}, equivalent elastic shear modulus for horizontal displacements.
3. G_{rr}, equivalent elastic shear modulus for rotational displacements.
4. ν, Poisson's ratio of the soil.

5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines for MODULI=SPUD CAN and NDIM=3:

First line:

1. G_{vv}, equivalent elastic shear modulus for vertical displacements.
2. G_{hh}, equivalent elastic shear modulus for horizontal displacements.
3. G_{rr}, equivalent elastic shear modulus for rotational displacements.
4. ν, Poisson's ratio of the soil.
5. k_t, torsional elastic modulus.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines for MODULI=GENERAL and NDIM=2:

First line:

1. k_{1111}.
2. k_{1122}.
3. k_{2222}.
4. k_{1112}.
5. k_{2212}.
6. k_{1212}.

7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

Data lines for MODULI=GENERAL and NDIM=3:

First line:

1. k_{1111}.
2. k_{1122}.
3. k_{2222}.
4. k_{1133}.
5. k_{2233}.
6. k_{3333}.
7. k_{1112}.
8. k_{2212}.

Second line:

1. k_{3312}.
2. k_{1212}.
3. k_{1113}.
4. k_{2213}.
5. k_{3313}.
6. k_{1213}.
7. k_{1313}.
8. k_{1123}.

Third line:

1. k_{2223}.
2. k_{3323}.
3. k_{1223}.
4. k_{1323}.
5. k_{2323}.
6. Temperature.

7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the elastic behavior as a function of temperature and other predefined field variables.

10.3 *JOINT PLASTICITY: Specify plastic properties for elastic-plastic joint elements.

This option is used to define the plastic behavior for elastic-plastic joint elements. It can be used only in conjunction with the *EPJOINT option.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Elastic-plastic joints," Section 29.11.1 of the Abaqus Analysis User's Manual
- *EPJOINT

Required parameter:

TYPE

Set TYPE=SAND to specify the model for interaction of spud cans and sand. Set TYPE=CLAY to specify the model for interaction of spud cans and clay. Set TYPE=MEMBER to specify the parabolic model for structural members.

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the plasticity property values. If this parameter is omitted, it is assumed that the plasticity property values are constant or depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines for TYPE=SAND:

First line:

1. V_t, yield strength in pure tension.
2. Λ_1, constant for yield surface shape.
3. Λ_2, constant for yield surface shape.
4. ϕ, soil friction angle, in degrees.
5. γ, soil unit weight.
6. Temperature.

7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the plastic behavior as a function of temperature and other predefined field variables.

Data lines for TYPE=CLAY:

First line:

1. s_u, undrained shear strength of the clay.
2. *a*, hardening parameter.
3. *b*, hardening parameter.
4. *c*, hardening parameter.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the plastic behavior as a function of temperature and other predefined field variables.

Data lines for TYPE=MEMBER:

First line:

1. V_c, compressive capacity.
2. V_t, tensile capacity.
3. H_m, horizontal capacity.
4. M_m, moment capacity.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the plastic behavior as a function of temperature and other predefined field variables.

10.4 *JOINTED MATERIAL: Specify the jointed material model.

This option is used to define a failure surface and the flow parameters for a single joint system or for bulk material failure in the elastic-plastic model of a jointed material, or it can be used to define shear retention in open joints. Up to three joint systems can be defined for each material point.

Product: Abaqus/Standard

Type: Model data

Level: Model

Reference:

- "Jointed material model," Section 20.5.1 of the Abaqus Analysis User's Manual

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the parameters of the model, in addition to temperature. If this parameter is omitted, it is assumed that the parameters depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

JOINT DIRECTION

Set this parameter equal to the name of the *ORIENTATION used to define the direction of a joint system. This use of the *ORIENTATION option does not affect the output of components of stress and strain—it only defines the joint orientation in the original configuration. Omit this parameter to give the bulk material failure parameters. The JOINT DIRECTION parameter cannot be used with the SHEAR RETENTION parameter.

NO SEPARATION

Include this parameter to prevent the joint from opening. This parameter must be used in conjunction with the JOINT DIRECTION parameter.

SHEAR RETENTION

Include this parameter to define shear retention in open joints. If this parameter is omitted, zero shear retention is assumed. The SHEAR RETENTION parameter cannot be used with the JOINT DIRECTION parameter.

Data lines defining failure surface and flow parameters (SHEAR RETENTION omitted):

First line:

1. Angle of friction, β, for this system. Give the value in degrees.
2. Dilation angle, ψ, for this system. Give the value in degrees.
3. Cohesion, d, for this system. (Units of FL^{-2}.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the failure and surface flow parameters on temperature and other predefined field variables.

Data lines defining the shear retention in open joints (SHEAR RETENTION included):

First line:

1. Fraction of elastic shear modulus retained when joints are open, f_{sr}. This value cannot be less than zero.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the shear retention on temperature and other predefined field variables.

10.5 *JOULE HEAT FRACTION: Define the fraction of electric energy released as heat.

This option is used to specify the fraction of dissipated electrical energy released as heat in coupled thermal-electrical problems.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Coupled thermal-electrical analysis," Section 6.7.2 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data line to define the joule heat fraction:

First (and only) line:

1. Fraction of electrical energy released as heat, including any unit conversion factor. The default value is 1.0.

11. K

11.1 *KAPPA: Specify the material parameters κ_s and κ_p for mass diffusion driven by gradients of temperature and equivalent pressure stress, respectively.

This option is used to introduce temperature- and pressure-driven mass diffusion. It must appear immediately after the *DIFFUSIVITY option. For each use of the *DIFFUSIVITY option, *KAPPA can be used once with TYPE=TEMP and once with TYPE=PRESS. The *KAPPA, TYPE=TEMP and *DIFFUSIVITY, LAW=FICK options are mutually exclusive.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Diffusivity," Section 23.5.1 of the Abaqus Analysis User's Manual
- *DIFFUSIVITY

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of κ_s or κ_p. If this parameter is omitted, κ_s or κ_p is assumed not to depend on any field variables but may still depend on concentration and temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

TYPE

Set TYPE=TEMP (default) to define κ_s (governing mass diffusion caused by temperature gradients). Set TYPE=PRESS to define κ_p (governing mass diffusion caused by gradients of the equivalent pressure stress).

Data lines to define the Soret effect factor (TYPE=TEMP):

First line:

1. Soret effect factor, κ_s. (Units of $\mathrm{F}^{1/2}\mathrm{L}^{-1}$.)
2. Concentration, c.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define κ_s as a function of concentration, temperature, and other predefined field variables.

Data lines to define the pressure stress factor (TYPE=PRESS):

First line:

1. Pressure stress factor, κ_p. (Units of $\mathrm{LF}^{-1/2}$.)
2. Concentration, c.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define κ_p as a function of concentration, temperature, and other predefined field variables.

11.2 *KINEMATIC: Define a kinematic coupling constraint.

This option is used to define a kinematic coupling constraint. It must be used in conjunction with the *COUPLING option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- "Coupling constraints," Section 31.3.2 of the Abaqus Analysis User's Manual
- *COUPLING

There are no parameters associated with this option.

Data lines to specify the degrees of freedom to be constrained:

First line:

1. First degree of freedom constrained. See "Conventions," Section 1.2.2 of the Abaqus Analysis User's Manual, for a definition of the numbering of degrees of freedom in Abaqus. If this field is left blank, all degrees of freedom will be constrained.
2. Last degree of freedom constrained. If this field is left blank, the degree of freedom specified in the first field will be the only one constrained.

Repeat this data line as often as necessary to specify constraints for different degrees of freedom. When the ORIENTATION parameter is specified on the associated *COUPLING option, the degrees of freedom are in the referenced local system in the initial configuration; otherwise, they are in the global system. In either case these directions will rotate with the reference node in large-displacement analyses (when the NLGEOM parameter is included on the *STEP option).

11.3 *KINEMATIC COUPLING: Constrain all or specific degrees of freedom of a set of nodes to the rigid body motion of a reference node.

This option is used to impose constraints between degrees of freedom of a node or node set and the rigid body motion defined by a reference node. The preferred method of providing a kinematic constraint of this type is the *COUPLING option used in conjunction with the *KINEMATIC option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Unsupported; this option has been superseded by coupling constraints used in conjunction with the kinematic option.

Reference:

- "Kinematic constraints: overview," Section 31.1.1 of the Abaqus Analysis User's Manual

Required parameter:

REF NODE

Set this parameter equal to either the node number of the reference node or the name of a node set containing the reference node. If the name of a node set is chosen, the node set must contain exactly one node.

Optional parameter:

ORIENTATION

Set this parameter equal to the name given to the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) that specifies the initial orientation of the local system in which the constrained degrees of freedom are defined.

Data lines to specify the nodes and degrees of freedom to be constrained:

First line:

1. Node number or node set label.
2. First degree of freedom constrained. See "Conventions," Section 1.2.2 of the Abaqus Analysis User's Manual, for a definition of the numbering of degrees of freedom in Abaqus/Standard. If this field is left blank, all degrees of freedom will be constrained.

3. Last degree of freedom constrained. If this field is left blank, the degree of freedom specified in the second field will be the only one constrained.

Repeat this data line as often as necessary to specify constraints at different nodes and degrees of freedom. When the ORIENTATION parameter is specified, the degrees of freedom are in the referenced local system in the initial configuration; otherwise, they are in the global system. In either case these directions will rotate with the reference node in large-displacement analyses (when the NLGEOM parameter is included on the *STEP option).

12. L

12.1 *LATENT HEAT: Specify latent heats.

This option is used to specify a material's latent heat.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Latent heat," Section 23.2.4 of the Abaqus Analysis User's Manual

Optional parameter:

PORE FLUID

This parameter applies only to Abaqus/Standard analyses.
Include this parameter if the latent heat of the pore fluid in a porous medium is being defined.

Data lines to define a material's latent heat:

First line:

1. Latent heat per unit mass. (Units of JM^{-1}.)
2. Solidus temperature.
3. Liquidus temperature.

Repeat this data line as often as necessary to define phase changes in the material; one line per phase change. Latent heat values must be given in ascending order of temperature.

12.2 *LOAD CASE: Begin a load case definition for multiple load case analysis.

This option is used to begin each load case definition.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- "Multiple load case analysis," Section 6.1.3 of the Abaqus Analysis User's Manual
- *END LOAD CASE

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the load case.

There are no data lines associated with this option.

12.3 *LOADING DATA: Provide loading data for uniaxial behavior models in connectors or provide data from a uniaxial or a shear loading test for fabric materials.

This option is used to define the loading response data for the uniaxial behavior of connector elements when used in conjunction with the *CONNECTOR BEHAVIOR and *CONNECTOR UNIAXIAL BEHAVIOR options.

This option is used to define the loading response from a uniaxial or a shear test for fabric materials when used in conjunction with the *FABRIC and *UNIAXIAL options. A fabric uniaxial test is specified with increasing strains along the specified yarn direction. A fabric shear test is specified with increasing shear strains as the fill and the warp yarns rotate with respect to each other.

If necessary, a separate unloading response can be specified using the *UNLOADING DATA option.

Product: Abaqus/Explicit

Type: Model data

Level: Model

Defining the loading response data for uniaxial behavior in connectors

References:

- "Connector uniaxial behavior," Section 28.2.10 of the Abaqus Analysis User's Manual
- *CONNECTOR BEHAVIOR
- *CONNECTOR UNIAXIAL BEHAVIOR
- *UNLOADING DATA

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the data, in addition to temperature. If this parameter is omitted, it is assumed that the data depend only on temperature. The DEPENDENCIES parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

DIRECTION

Set DIRECTION=TENSION to define tensile behavior.

Set DIRECTION=COMPRESSION to define compressive behavior.

If this parameter is omitted, the behavior is assumed to be nonlinear elastic and the data may span both positive and negative values of the primary component. The behavior will be considered

to be symmetric about the origin if the given data are limited to either positive or negative values of the primary component.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

The default is EXTRAPOLATION=CONSTANT unless *CONNECTOR BEHAVIOR, EXTRAPOLATION=LINEAR is used. The EXTRAPOLATION parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

INDEPENDENT COMPONENTS

Set INDEPENDENT COMPONENTS=CONSTITUTIVE MOTION to specify dependencies on components of constitutive relative motion included in the uniaxial behavior definition. The INDEPENDENT COMPONENTS parameter should not be used if uniaxial behavior is dependent on only the component of constitutive relative motion specified with the COMPONENT parameter on the *CONNECTOR UNIAXIAL BEHAVIOR option. The INDEPENDENT COMPONENTS parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

REGULARIZE

Set REGULARIZE=ON to regularize the user-defined tabular loading data.

Set REGULARIZE=OFF to use the user-defined tabular loading data directly without regularization.

The default is REGULARIZE=ON unless *CONNECTOR BEHAVIOR, REGULARIZE=OFF is used. The REGULARIZE parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

RTOL

Set this parameter equal to the tolerance to be used to regularize the user-defined tabular loading data. If this parameter is omitted, the default is RTOL=0.03 unless the tolerance is specified on the *CONNECTOR BEHAVIOR option.

This parameter is ignored if REGULARIZE=OFF.

The RTOL parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

TYPE

Set TYPE=DAMAGE to define a damage elasticity model. TYPE=DAMAGE must be used in conjunction with the DIRECTION parameter.

Set TYPE=ELASTIC (default) to define a nonlinear elastic model with or without rate dependency.

Set TYPE=PERMANENT DEFORMATION to define models that exhibit permanent deformation (plasticity) upon unloading. TYPE=PERMANENT DEFORMATION must be used in conjunction with the DIRECTION parameter.

Rate-independent elastic models do not require the definition of unloading data. Nonelastic models and rate-dependent models require the specification of unloading behavior using the *UNLOADING DATA option.

The following parameters are optional and can be used only with TYPE=ELASTIC:

RATE DEPENDENT

Include this parameter equal to define rate-dependent loading data. If this parameter is omitted, the data are assumed to be rate independent.

RATE INTERPOLATION

Set RATE INTERPOLATION=LINEAR (default) to use linear intervals for the relative motion rate while interpolating rate-dependent loading data.

Set RATE INTERPOLATION=LOGARITHMIC to use logarithmic intervals for the equivalent relative motion rate while interpolating rate-dependent loading data.

This parameter is ignored if the RATE DEPENDENT parameter is omitted.

The following parameter is optional and can be used only with TYPE=DAMAGE:

DAMAGE ONSET

Set this parameter equal to the displacement/strain value at which the onset of damage occurs.

The following parameters are optional, mutually exclusive, and can be used only with TYPE=PERMANENT DEFORMATION:

SLOPE DROP

Set this parameter equal to the relative drop in slope on the loading curve that defines the onset of plastic deformation. The default value is 0.1.

YIELD ONSET

Set this parameter equal to the displacement/strain value at which the onset of yield occurs.

Data lines to define uniaxial behavior that depends on the displacement/rotation in the direction of the specified component of relative motion (the RATE DEPENDENT and INDEPENDENT COMPONENTS parameters are omitted):

First line:

1. Force or moment. (Provide the absolute value if the DIRECTION parameter is defined.)
2. Constitutive relative displacement or rotation. (Provide the absolute value if the DIRECTION parameter is defined.)
3. Temperature.

4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the loading curve data.

Data lines to define uniaxial behavior that depends on the relative positions or motions in several component directions (the RATE DEPENDENT parameter is omitted and the INDEPENDENT COMPONENTS parameter is included):

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).
3. Etc., up to N_i entries. The independent components should not include the component for which the uniaxial loading behavior is being defined.

Subsequent lines:

1. Force or moment in the direction specified by the COMPONENT parameter on the *CONNECTOR UNIAXIAL BEHAVIOR option. (Provide the absolute value if the DIRECTION parameter is defined.)
2. Constitutive relative displacement or rotation. (Provide the absolute value if the DIRECTION parameter is defined.)
3. Constitutive relative motion in the first independent component identified on the first data line.
4. Constitutive relative motion in the second independent component identified on the first data line.
5. Etc., up to N_i entries as identified on the first data line.
6. Temperature.
7. First field variable.
8. Second field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the loading curve data.

Data lines to define rate-dependent uniaxial behavior in the direction of the specified component of relative motion (the RATE DEPENDENT parameter is included and the INDEPENDENT COMPONENTS parameter is omitted):

First line:

1. μ_0.
2. μ_1.
3. α.

Second line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Relative velocity. (Provide the absolute value.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the loading curve data.

Data lines to define rate-dependent uniaxial behavior that depends on the relative motions in several component directions (the RATE DEPENDENT and INDEPENDENT COMPONENTS parameters are included):

First line:

1. First independent component number (1–6).
2. Second independent component number (1–6).
3. Etc., up to N_i entries. The independent components should not include the component for which the uniaxial loading behavior is being defined.

Second line:

1. μ_0.
2. μ_1.
3. α.

Subsequent lines:

1. Force or moment in the direction specified by the COMPONENT parameter on the *CONNECTOR UNIAXIAL BEHAVIOR option. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Relative velocity. (Provide the absolute value.)
4. Constitutive relative motion in the first independent component identified on the first data line.
5. Etc., up to N_i entries as identified on the first data line.
6. Temperature.
7. First field variable.
8. Etc., up to eight entries per line.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first and second data lines. Repeat the subsequent data lines as often as necessary to define the loading curve data.

Defining the loading response data from uniaxial tests of fabric materials

References:

- "Fabric material behavior," Section 20.4.1 of the Abaqus Analysis User's Manual
- *FABRIC
- *UNIAXIAL
- *UNLOADING DATA

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the data, in addition to temperature. If this parameter is omitted, it is assumed that the data depend only on temperature. The DEPENDENCIES parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

DIRECTION

Set DIRECTION=TENSION for tests with the primary strain component positive.

Set DIRECTION=COMPRESSION for tests with the primary strain component negative.

If this parameter is omitted, the behavior is assumed to be nonlinear elastic and the data may span both positive and negative values of the primary strain component. The behavior will be considered to be symmetric about the origin if the given data are limited to either positive or negative values of the primary strain component.

EXTRAPOLATION

Set EXTRAPOLATION=CONSTANT to use constant extrapolation of the dependent variables outside the specified range of the independent variables.

Set EXTRAPOLATION=LINEAR (default) to use linear extrapolation of the dependent variables outside the specified range of the independent variables.

The EXTRAPOLATION parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

REGULARIZE

Set REGULARIZE=ON (default) to regularize the user-defined tabular loading data.

Set REGULARIZE=OFF to use the user-defined tabular loading data directly without regularization.

The REGULARIZE parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

RTOL

Set this parameter equal to the tolerance to be used to regularize the user-defined tabular loading data. If this parameter is omitted, the default is RTOL=0.03.

This parameter is ignored if REGULARIZE=OFF.

The RTOL parameter also applies to any subsequent unloading data defined using the *UNLOADING DATA option.

TYPE

Set TYPE=DAMAGE to define a damage elasticity model. TYPE=DAMAGE must be used in conjunction with the DIRECTION parameter.

Set TYPE=ELASTIC (default) to define a nonlinear elastic model with or without rate dependency.

Set TYPE=PERMANENT DEFORMATION to define models that exhibit permanent deformation (plasticity) upon unloading. TYPE=PERMANENT DEFORMATION must be used in conjunction with the DIRECTION parameter.

Rate-independent elastic models do not require the definition of unloading data. Nonelastic models and rate-dependent models require the specification of unloading test data using the *UNLOADING DATA option.

The following parameters are optional and can be used only with TYPE=ELASTIC:

RATE DEPENDENT

Include this parameter equal to define rate-dependent loading data. If this parameter is omitted, the data are assumed to be rate independent.

RATE INTERPOLATION

Set RATE INTERPOLATION=LINEAR (default) to use linear intervals for strain rate while interpolating rate-dependent loading data.

Set RATE INTERPOLATION=LOGARITHMIC to use logarithmic intervals for strain rate while interpolating rate-dependent loading data.

This parameter is ignored if the RATE DEPENDENT parameter is omitted.

The following parameter is optional and can be used only with TYPE=DAMAGE:

DAMAGE ONSET

Set this parameter equal to the displacement/strain value at which the onset of damage occurs.

The following parameters are optional, mutually exclusive, and can be used only with TYPE=PERMANENT DEFORMATION:

SLOPE DROP

Set this parameter equal to the relative drop in slope on the loading curve that defines the onset of plastic deformation. The default value is 0.1.

YIELD ONSET

Set this parameter equal to the strain value at which the onset of yield occurs.

Data lines to define rate-independent fabric response (the RATE DEPENDENT parameter is omitted):

First line:

1. Nominal stress (Provide the absolute value if the DIRECTION parameter is defined.)
2. Nominal strain. (Provide the absolute value if the DIRECTION parameter is defined.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the loading curve data.

Data lines to define rate-dependent fabric response (the RATE DEPENDENT parameter is included):

First line:

1. μ_0.
2. μ_1.
3. α.

Second line:

1. Nominal stress. (Provide the absolute value.)
2. Nominal strain. (Provide the absolute value.)
3. Strain rate. (Provide the absolute value.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the second and subsequent data lines as often as necessary to define the loading curve data.

12.4 *LOW DENSITY FOAM: Specify properties for a low-density foam.

This option is used to define material coefficients for low-density foam materials. The *LOW DENSITY FOAM option must be used in conjunction with the *UNIAXIAL TEST DATA, DIRECTION=TENSION and the *UNIAXIAL TEST DATA, DIRECTION=COMPRESSION options to specify the stress-strain response of the foam material in uniaxial tension and compression, respectively.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Low-density foams," Section 19.9.1 of the Abaqus Analysis User's Manual
- *UNIAXIAL TEST DATA

Optional parameters:

STRAIN RATE

Use this parameter to define the strain rate measure used for constitutive calculations.

Set STRAIN RATE=VOLUMETRIC (default) to use the volumetric strain rate.

Set STRAIN RATE=PRINCIPAL to use the nominal strain rate along each principal direction of deformation.

RATE EXTRAPOLATION

Use this parameter to specify extrapolation of rate-dependent uniaxial stress-strain curves beyond the maximum specified strain rate.

Set RATE EXTRAPOLATION=NO (default) to prevent extrapolation. The curve corresponding to the maximum specified strain rate will be used for strain rates greater than the maximum.

Set RATE EXTRAPOLATION=YES to trigger extrapolation beyond the maximum specified strain rate by using the slope with respect to strain rate.

TENSION CUTOFF

Include this parameter to specify a tension cutoff value for the maximum principal stress that the low-density material can sustain with tension. The tension cutoff value must be greater than zero.

FAIL

This parameter is relevant only when the TENSION CUTOFF parameter is used.

Set FAIL=NO (default) to force the maximum principal stress to remain below the tension cutoff without deleting the element.

Set FAIL=YES to allow element deletion when the tension cutoff value is reached.

Data line to specify the relaxation coefficients for low-density foams:

First (and only) line:

1. μ_0. The default is =0.0001 (time units).
2. μ_1. The default is =0.005 (time units).
3. α. The default is 2.

13. M

13.1 *MAP SOLUTION: Map a solution from an old mesh to a new mesh.

This option is used to transfer solution variables from an earlier analysis to a new mesh that occupies the same space.

Product: Abaqus/Standard

Type: Model data

Level: Model

Reference:

- "Mesh-to-mesh solution mapping," Section 12.4.1 of the Abaqus Analysis User's Manual

Optional parameters:

INC

Set this parameter equal to the increment number from which the old solution will be read. If this parameter is omitted, the last increment for which a solution is available will be read.

The STEP parameter must be specified if the INC parameter is used.

STEP

Set this parameter equal to the step number from which the old solution will be read. If this parameter is omitted, the last step and increment for which a solution is available will be read.

UNBALANCED STRESS

Set UNBALANCED STRESS=RAMP (default) if the stress unbalance is to be resolved linearly over the step.

Set UNBALANCED STRESS=STEP if the stress unbalance is to be resolved in the first increment.

Data line to translate an old-model mesh:

First (and only) line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Data lines to translate and/or rotate an old-model mesh:

First line:

1. Value of the translation to be applied in the *X*-direction.

2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Enter values of zero to apply a pure rotation.

Second line:

1. *X*-coordinate of point *a* on the axis of rotation (see Figure 13.1–1).
2. *Y*-coordinate of point *a* on the axis of rotation.
3. *Z*-coordinate of point *a* on the axis of rotation.
4. *X*-coordinate of point *b* on the axis of rotation.
5. *Y*-coordinate of point *b* on the axis of rotation.
6. *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

If both translation and rotation are specified, translation is applied before rotation.

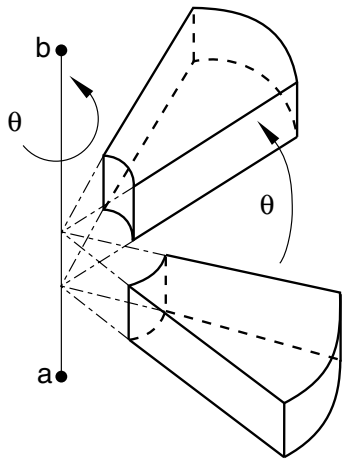


Figure 13.1–1 Rotation of an old-model mesh.

13.2 *MASS: Specify a point mass.

This option is used to define lumped mass values associated with MASS elements.

For Abaqus/Standard analyses this option is also used to define mass proportional damping (for direct-integration dynamic analysis) and composite damping (for modal dynamic analysis) associated with MASS elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Property module and Interaction module.

Reference:

- "Point masses," Section 27.1.1 of the Abaqus Analysis User's Manual

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the MASS elements for which the value is being given.

Optional parameters:

ALPHA

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the α_R factor to create mass proportional damping for the MASS elements when used in direct-integration dynamics. This value is ignored in modal dynamics. The default is 0.0.

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the fraction of critical damping to be used with the MASS elements when calculating composite damping factors for the modes when used in modal dynamics. This value is ignored in direct-integration dynamics. The default is 0.0.

Data line to define the mass magnitude:

First (and only) line:

1. Mass magnitude. Mass, not weight, should be given.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

13.3 *MASS DIFFUSION: Transient or steady-state uncoupled mass diffusion analysis.

This option is used to control uncoupled transient or steady-state mass diffusion analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

Reference:

- "Mass diffusion analysis," Section 6.9.1 of the Abaqus Analysis User's Manual

Optional parameters:

DCMAX

Set this parameter equal to the maximum normalized concentration change to be allowed in an increment. Abaqus/Standard will restrict the time step to ensure that this value will not be exceeded at any node (except nodes with boundary conditions) during any increment of the analysis. If the DCMAX parameter is omitted, fixed time increments will be used.

END

Set END=PERIOD (default) to analyze the entire time period specified on the data line. Set END=SS to end the analysis when steady state is reached.

STEADY STATE

Include this parameter to choose steady-state analysis. Transient analysis is assumed if this parameter is omitted.

Data line to define time stepping in a mass diffusion analysis:

First (and only) line:

1. Time step. If automatic stepping is used, this value should be a reasonable suggestion for the initial step and will be adjusted as necessary.
2. Time period. If END=SS is chosen, the step ends when steady state is reached or after this time period, whichever occurs first.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If no value is given, Abaqus/Standard sets the minimum increment to the minimum of 0.8 times the suggested initial time step (the first data item on this line) and 10^{-5} times the time period (the second data item on this line). If a value is

given, Abaqus/Standard will use the minimum of the given value and 0.8 times the suggested initial time step.

4. Maximum time increment. If this value is omitted, no upper limit is imposed. This value is used only for automatic time incrementation.
5. Rate of change of normalized concentration (normalized concentration per time) used to define steady state; only needed if END=SS is chosen. When all nodal normalized concentrations are changing at less than this rate, the solution terminates.

13.4 *MASS FLOW RATE: Specify fluid mass flow rate in a heat transfer analysis.

This option is used to specify the mass flow rate per unit area (or through the entire section for one-dimensional elements) for forced convection/diffusion elements in a heat transfer analysis. This option cannot be used with hydrostatic fluid elements.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Uncoupled heat transfer analysis," Section 6.5.2 of the Abaqus Analysis User's Manual
- "UMASFL," Section 1.1.35 of the Abaqus User Subroutines Reference Manual

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude versus time curve that defines the magnitude of the flow rate during the step ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual). If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

OP

Set OP=MOD (default) for existing *MASS FLOW RATE values to remain, with this option modifying existing flow rates or defining additional flow rates.

Set OP=NEW if all existing *MASS FLOW RATE values applied to the model should be removed.

USER

Include this parameter to indicate that user subroutine **UMASFL** will be used to define mass flow rate values. **UMASFL** will be called for each node given on the data lines. If values are also given on the data lines, these values will be ignored.

Data lines to define mass flow rates:

First line:

1. Node number or node set label.
2. Mass flow rate per unit area in the x-direction (units of $ML^{-2}T^{-1}$) or total mass flow rate in the cross-section (units of MT^{-1}) for one-dimensional elements.
3. Mass flow rate per unit area in the y-direction (not needed for nodes associated with one-dimensional elements).
4. Mass flow rate per unit area in the z-direction (not needed for nodes associated with one-dimensional elements).

Repeat this data line as often as necessary to define mass flow rates at different nodes.

Data lines to define mass flow rates using user subroutine `UMASFL`:

First line:

1. Node set or node number.

Repeat this data line as often as necessary. **`UMASFL`** will be called for each node listed.

13.5 *MATERIAL: Begin the definition of a material.

This option is used to indicate the start of a material definition.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the material in the element property options. Material names in the same input file must be unique. Furthermore, material names should be unique from the names associated with property definitions such as *CONNECTOR BEHAVIOR and *FLUID BEHAVIOR. Material names adhere to the naming convention for labels (see "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual), except that they cannot begin with a number.

Optional parameters:

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used for regularizing the material data. The default is RTOL=0.03.

SRATE FACTOR

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the factor used for filtering the equivalent plastic strain rate for the evaluation of strain rate-dependent material data. The default value is 0.9.

STRAIN RATE REGULARIZATION

This parameter applies only to Abaqus/Explicit analyses and is used only to regularize strain rate-dependent material data.

Set STRAIN RATE REGULARIZATION=LOGARITHMIC (default) to use a logarithmic regularization for strain rate-dependent material data.

Set STRAIN RATE REGULARIZATION=LINEAR to use a linear regularization for strain rate-dependent material data.

There are no data lines associated with this option.

13.6 *MATRIX: Read in the stiffness or mass matrix for a linear user element.

This option can be used only in conjunction with the *USER ELEMENT, LINEAR option. It is used to read in the stiffness or mass matrix for the user element. It can be used once if only a stiffness or mass is required or twice to give both matrices.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance, Model

References:

- "User-defined elements," Section 29.16.1 of the Abaqus Analysis User's Manual
- *USER ELEMENT

Required parameter:

TYPE

Set TYPE=MASS to define the mass matrix. Set TYPE=STIFFNESS to define the stiffness matrix.

Optional parameter:

INPUT

Set this parameter equal to the name of the alternate input file from which the data lines are to be read. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define the matrix:

First line:

1. Matrix entries, four per line.

Repeat this data line as often as necessary.

13.7 *MATRIX ASSEMBLE: Define stiffness, mass, or damping matrices for a part of the model.

This option can be used to identify a stiffness, mass, or damping matrix that will be assembled into the corresponding global finite element matrix. This matrix must have been input previously by using the *MATRIX INPUT option.

Product: Abaqus/Standard

Type: Model data

Level: Model

References:

- "Defining matrices," Section 2.10.1 of the Abaqus Analysis User's Manual
- *MATRIX INPUT

At least one of the following parameters is required:

MASS

Set this parameter equal to the name of the mass matrix.

STIFFNESS

Set this parameter equal to the name of the stiffness matrix.

STRUCTURAL DAMPING

Set this parameter equal to the name of the structural damping matrix.

VISCOUS DAMPING

Set this parameter equal to the name of the viscous damping matrix.

There are no data lines associated with this option.

13.8 *MATRIX GENERATE: Generate global matrices.

This option is used to generate global matrices representing the stiffness, mass, damping, or load vectors in a model.

Product: Abaqus/Standard

Type: History data

Level: Step

Reference:

- "Generating global matrices," Section 10.3.1 of the Abaqus Analysis User's Manual

At least one of the following parameters is required:

STIFFNESS

Include this parameter to generate the stiffness matrix.

MASS

Include this parameter to generate the mass matrix.

VISCOUS DAMPING

Include this parameter to generate the viscous damping matrix.

STRUCTURAL DAMPING

Include this parameter to generate the structural damping matrix.

LOAD

Include this parameter to generate the load matrix.

Optional parameter:

PROPERTY EVALUATION

Set this parameter equal to the frequency at which to evaluate frequency-dependent properties for viscoelasticity, springs, and dashpots during the matrix generation. If this parameter is omitted, Abaqus/Standard will evaluate the stiffness associated with frequency-dependent springs and dashpots at zero frequency and will not consider the stiffness contributions from frequency-domain viscoelasticity.

There are no data lines associated with this option.

13.9 *MATRIX INPUT: Read in a matrix for a part of the model.

This option can be used to input a matrix in sparse format.

Product: Abaqus/Standard

Type: Model data

Level: Model

References:

- "Defining matrices," Section 2.10.1 of the Abaqus Analysis User's Manual
- *MATRIX ASSEMBLE

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to this matrix.

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file from which the data lines are to be read. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

SCALE FACTOR

Set this parameter equal to a nonzero real number s by which all matrix entries will be multiplied. The default value is $s = 1.0$.

TYPE

This parameter defines the shape of the matrix.

Set TYPE=SYMMETRIC (default) to read the upper or lower triangular portion of a square symmetric matrix. If a full matrix is specified, corresponding terms above and below the diagonal must be equal.

Set TYPE=UNSYMMETRIC to read a square unsymmetric matrix.

Data lines to define the matrix in sparse format (only nonzero terms):

First line:

1. Row node number.
2. Degree of freedom number for row node.

3. Column node number.
4. Degree of freedom number for column node.
5. Matrix entry.

Give data to define a symmetric matrix in lower triangular, upper triangular, or square format. For a square matrix to be symmetric, corresponding entries above and below the diagonal must have exactly the same values.

Repeat this data line as often as necessary.

13.10 *MEMBRANE SECTION: Specify section properties for membrane elements.

This option is used to assign section properties to a set of membrane elements. Section properties include thickness, thickness change behavior, material definition, and material orientation.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

Reference:

- "Membrane elements," Section 26.1.1 of the Abaqus Analysis User's Manual

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the membrane elements for which the section properties are being defined.

MATERIAL

Set this parameter equal to the name of the material to be used with these elements.

Optional parameters:

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see "Section controls," Section 24.1.4 of the Abaqus Analysis User's Manual) to be used to specify a nondefault hourglass control formulation option or scale factors.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition to be used to specify the enhanced hourglass control formulation (see "Section controls," Section 24.1.4 of the Abaqus Analysis User's Manual) or to be used in a subsequent Abaqus/Explicit import analysis.

DENSITY

Set this parameter equal to a mass per unit surface area of the membrane.

If this parameter is used, the mass of the membrane includes a contribution from this parameter in addition to any contribution from the material definition.

NODAL THICKNESS

Include this parameter to indicate that the membrane thickness should not be read from the data lines but should be interpolated from the thickness specified at the nodes with the *NODAL THICKNESS option.

ORIENTATION

Set this parameter equal to the name given for the *ORIENTATION option to be used to define a local coordinate system for material calculations in the elements in this set.

POISSON

This parameter is relevant only in a large-deformation analysis. Set it equal to a nonzero value to cause the thickness to change as a function of membrane strains. The value of the POISSON parameter must be between −1.0 and 0.5. A value of 0.5 will enforce incompressible behavior of the element. POISSON=0.0 means that the thickness will not change.

Set this parameter equal to MATERIAL in an Abaqus/Explicit analysis to cause the thickness to change based on the element material definition.

The default is POISSON=0.5 in Abaqus/Standard and POISSON=MATERIAL in Abaqus/Explicit.

Data line for a constant thickness membrane:

First (and only) line:

1. Section thickness.

To define a continuously varying thickness membrane:

No data lines are used with this option when the NODAL THICKNESS parameter is specified; any value input on the data line will be ignored. Instead, the *NODAL THICKNESS option is used to define the section thickness.

13.11 *MODAL DAMPING: Specify damping for modal dynamic analysis.

This option is used to specify damping for mode-based procedures. It is usually used in conjunction with the *SELECT EIGENMODES option for selecting eigenmodes for modal superposition. If the *SELECT EIGENMODES option is not used, all eigenmodes extracted in the prior *FREQUENCY step will be used with the damping values specified under the *MODAL DAMPING option. If the *MODAL DAMPING option is not used, zero damping values are assumed.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Material damping," Section 23.1.1 of the Abaqus Analysis User's Manual
- "Dynamic analysis procedures: overview," Section 6.3.1 of the Abaqus Analysis User's Manual

Optional, mutually exclusive parameters:

MODAL

Set MODAL=DIRECT to select modal damping using the damping coefficients given in this option. The data lines after the keyword line specify the modal damping values to be used in the analysis. If the *MODAL DAMPING option is used without parameters, MODAL=DIRECT is assumed.

Set MODAL=COMPOSITE to select composite modal damping using the damping coefficients that have been calculated in the *FREQUENCY step ("Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Manual) from the material damping factors given on the *DAMPING material definition option ("Material damping," Section 23.1.1 of the Abaqus Analysis User's Manual). Composite modal damping can be used only with DEFINITION=MODE NUMBERS.

RAYLEIGH

Include this parameter to select Rayleigh damping. The damping term for a particular mode is defined as $\alpha_M m_M + \beta_M k_M$, where α_M and β_M are factors defined on the first data line of the option and m_M is the modal mass and k_M is the modal stiffness for mode *M*.

STRUCTURAL

Include this parameter to select structural damping, which means that the damping is proportional to the internal forces but opposite in direction to the velocity. This parameter can be used only with the *STEADY STATE DYNAMICS, *RANDOM RESPONSE, or SIM-based *MODAL DYNAMIC

procedures (see "Mode-based steady-state dynamic analysis," Section 6.3.8 of the Abaqus Analysis User's Manual; "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Manual; and "Transient modal dynamic analysis," Section 6.3.7 of the Abaqus Analysis User's Manual). The value of the damping constant, s, that multiplies the internal forces is entered on the data line.

Optional parameters:

DEFINITION

Set DEFINITION=MODE NUMBERS (default) to indicate that the damping values are given for the specified mode numbers.

Set DEFINITION=FREQUENCY RANGE to indicate that the damping values are given for the specified frequency ranges. Frequency ranges can be discontinuous.

If both the *MODAL DAMPING and *SELECT EIGENMODES options are used in the same step, the DEFINITION parameter must be set equal to the same value in both options.

FIELD

Set FIELD=ALL (default) to indicate that the damping values are to be applied to both structural and acoustic modes

Set FIELD=MECHANICAL to indicate that the damping values are to be applied only to structural modes.

Set FIELD=ACOUSTIC to indicate that the damping values are to be applied only to acoustic modes.

This option can be used only with MODAL=DIRECT and DEFINITION=FREQUENCY RANGE for uncoupled structural and acoustic modes obtained through AMS eigenextraction.

Data lines to define a fraction of critical damping by specifying mode numbers (MODAL=DIRECT and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Fraction of critical damping, ξ.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define Rayleigh damping by specifying mode numbers (RAYLEIGH and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)

3. Mass proportional damping, α_M.
4. Stiffness proportional damping, β_M.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define composite modal damping (MODAL=COMPOSITE):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define structural damping by specifying mode numbers (STRUCTURAL and DEFINITION=MODE NUMBERS):

First line:

1. Mode number of the lowest mode of a range.
2. Mode number of the highest mode of a range. (If this entry is left blank, it is assumed to be the same as the previous entry so that values are being given for one mode only.)
3. Damping factor, *s*.

Repeat this data line as often as necessary to define modal damping for different modes.

Data lines to define a fraction of critical damping by specifying frequency ranges (MODAL=DIRECT and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Fraction of critical damping, ξ.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus will interpolate linearly between frequencies and keep the damping value constant and equal to the closest specified value outside the frequency range.

Data lines to define Rayleigh damping by specifying frequency ranges (RAYLEIGH and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Mass proportional damping, α_M.

3. Stiffness proportional damping, β_M.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus will interpolate linearly between frequencies and keep the damping value constant and equal to the closest specified value outside the frequency range.

Data lines to define structural damping by specifying frequency ranges (STRUCTURAL and DEFINITION=FREQUENCY RANGE):

First line:

1. Frequency value (in cycles/time).
2. Damping factor, *s*.

Repeat this data line as often as necessary to define modal damping for different frequencies. Abaqus will interpolate linearly between frequencies and keep the damping value constant and equal to the closest specified value outside the frequency range.

13.12 *MODAL DYNAMIC: Dynamic time history analysis using modal superposition.

This option is used to provide dynamic time history response as a linear perturbation procedure using modal superposition.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

Reference:

- "Transient modal dynamic analysis," Section 6.3.7 of the Abaqus Analysis User's Manual

Optional parameter:

CONTINUE

Set CONTINUE=NO (default) to specify that this step is not to carry over the initial conditions from the results of the preceding step. In this case the initial displacements are zero, and the initial velocities are taken from the *INITIAL CONDITIONS, TYPE=VELOCITY option if it is used; otherwise, they are zero. Step time begins at zero.

Set CONTINUE=YES to specify that this step is to carry over the initial conditions from the end of the immediately preceding *MODAL DYNAMIC step or static perturbation step. If this preceding step is a *MODAL DYNAMIC step, both the velocities and the displacements from the end of this step are used as the initial conditions for the current step. If this preceding step is a static perturbation step, the displacements from this step are used as the initial displacements for the current step and the initial velocities are taken from the *INITIAL CONDITIONS, TYPE=VELOCITY option if it is used; otherwise, they are zero. Step time is continued from the immediately preceding *MODAL DYNAMIC or static perturbation step.

Data line for a transient modal dynamic analysis:

First (and only) line:

1. Time increment to be used.
2. Time period.

13.13 *MODAL FILE: Write generalized coordinate (modal amplitude) data or eigendata to the results file during a mode-based dynamic or eigenvalue extraction procedure.

This option is used during mode-based dynamic or eigenvalue extraction procedures to control the writing of generalized coordinate (modal amplitude and phase) values or eigendata to the Abaqus/Standard results file.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

Reference:

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Manual

Optional parameter:

FREQUENCY

This parameter is valid only in mode-based dynamic procedures.

Set this parameter equal to the output frequency, in increments. The output will always be written to the results file at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

Data lines to request modal output in the results file during mode-based dynamic procedures:

First line:

1. Give the identifying keys for the variables to be written to the results file. The keys are defined in the "Modal variables" section of "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary.

To write eigendata during an eigenvalue extraction procedure:

No data lines are required; the eigendata are written automatically.

13.14 *MODAL OUTPUT: Write generalized coordinate (modal amplitude) data to the output database during a mode-based dynamic or complex eigenvalue extraction procedure.

This option is used during a mode-based dynamic or complex eigenvalue extraction procedure to write generalized coordinate (modal amplitude and phase) values to the Abaqus/Standard output database. It must be used in conjunction with the *OUTPUT, HISTORY option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *OUTPUT

Optional parameter:

VARIABLE

Set VARIABLE=ALL to indicate that all modal variables applicable to this procedure and material type should be written to the output database.

If this parameter is omitted, the modal variables requested for output must be specified on the data lines.

Data lines to request modal output:

First line:

1. Give the identifying keys for the variables to be written to the output database. The keys are defined in the "Modal variables" section of "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary to define the modal variables to be written to the output database.

13.15 *MODAL PRINT: Print generalized coordinate (modal amplitude) data during a mode-based dynamic procedure.

This option is used during mode-based dynamic procedures to control the printed output of generalized coordinate (modal amplitude and phase) values.

Product: Abaqus/Standard

Type: History data

Level: Step

Reference:

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Manual

Optional parameter:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

Data lines to request modal output in the data file:

First line:

1. Give the identifying keys for the variables to be printed. The keys are defined in the "Modal variables" section of "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary: each line defines a table.

13.16 *MODEL CHANGE: Remove or reactivate elements and contact pairs.

This option is used to remove or reactivate elements or contact pairs during an analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Element and contact pair removal and reactivation," Section 11.2.1 of the Abaqus Analysis User's Manual
- "Removing and reactivating contact pairs" in "Defining contact pairs in Abaqus/Standard," Section 32.3.1 of the Abaqus Analysis User's Manual

Required, mutually exclusive parameters:

ACTIVATE

Include this parameter in any step during an analysis to indicate that elements or contact pairs may need to be removed or added during a subsequent restart analysis.

ADD

Include this parameter to indicate that the elements or contact pairs involved are being reactivated during the step.

Set ADD=STRAIN FREE (or include the ADD parameter without a value) to specify strain-free reactivation for stress/displacement elements or to reactivate other elements or contact pairs.

Set ADD=WITH STRAIN to specify that stress/displacement elements are reactivated with strain. This option is not relevant for contact pairs.

REMOVE

Include this parameter to indicate that the elements or contact pairs involved are being removed during the step.

Optional parameter:

TYPE

This parameter can be used only with the parameters ADD or REMOVE.

Set TYPE=ELEMENT (default) to remove or reactivate elements. Set TYPE=CONTACT PAIR to remove or reactivate contact pairs.

Data lines to remove/reactivate elements (TYPE=ELEMENT):

First line:

1. Give a list of element numbers and/or element set names that are involved in the removal or reactivation.

Repeat this data line as often as necessary.

Data lines to remove/reactivate contact pairs (TYPE=CONTACT PAIR):

First line:

1. Slave surface name used in the contact pair being removed or reactivated.
2. Master surface name used in the contact pair being removed or reactivated. For self-contact the master surface name is omitted or is the same as the slave surface name.

Repeat this data line as often as necessary.

No data lines are used with this option when the ACTIVATE parameter is included.

13.17 *MOHR COULOMB: Specify the Mohr-Coulomb plasticity model.

This option is used to define the yield surface and flow potential parameters for elastic-plastic materials that use the Mohr-Coulomb plasticity model. It must be used in conjunction with the *MOHR COULOMB HARDENING option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Mohr-Coulomb plasticity," Section 20.3.3 of the Abaqus Analysis User's Manual
- *MOHR COULOMB HARDENING

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material parameters other than temperature. If this parameter is omitted, it is assumed that the material properties are constant or depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

DEVIATORIC ECCENTRICITY

Set this parameter equal to the flow potential eccentricity in the deviatoric plane, e. This feature allows the shape of flow potential in the deviatoric stress space to be controlled independently of the angle of friction. If this parameter is omitted, the deviatoric eccentricity is calculated by default as $e = (3 - \sin\phi)/(3 + \sin\phi)$, where ϕ is the Mohr-Coulomb angle of friction defined on the data lines. The range of values e can have is $1/2 < e \leq 1$.

ECCENTRICITY

Set this parameter equal to the flow potential eccentricity in the meridional plane, ϵ. The meridional eccentricity is a small positive number that defines the rate at which the flow potential approaches its asymptote. The default is $\epsilon = 0.1$.

Data lines to define a Mohr-Coulomb plasticity model:

First line:

1. Friction angle, ϕ, at high confining pressure in the p–$R_{mc}q$ plane. Give the value in degrees. The friction angle can range from $0° \leq \phi \leq 89.9°$.
2. Dilation angle, ψ, at high confining pressure in the p–$R_{mw}q$ plane. Give the value in degrees. The dilation angle can range from $0° \leq \psi \leq 89.9°$. Abaqus will set ψ to $0.1°$ when $\phi > 0$ and the dilation angle is not specified or is specified as zero.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other predefined field variables.

13.18 *MOHR COULOMB HARDENING: Specify hardening for the Mohr-Coulomb plasticity model.

This option is used to define piecewise linear hardening/softening behavior for a material defined by the Mohr-Coulomb plasticity model. It must be used in conjunction with the *MOHR COULOMB option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Mohr-Coulomb plasticity," Section 20.3.3 of the Abaqus Analysis User's Manual
- *MOHR COULOMB

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the cohesion yield stress, in addition to temperature. If this parameter is omitted, it is assumed that the cohesion yield stress depends only on the plastic strain and, possibly, on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define Mohr-Coulomb hardening:

First line:

1. Cohesion yield stress.
2. Absolute value of the corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the cohesion yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

13.19 *MOISTURE SWELLING: Define moisture-driven swelling.

This option is used to define the moisture-driven swelling of the solid skeleton in a partially saturated porous medium. It can be used in the analysis of coupled wetting liquid flow and porous medium stress.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Moisture swelling," Section 23.7.6 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data lines to define moisture-driven swelling:

First line:

1. Volumetric moisture swelling strain, ε^{ms}.
2. Saturation, s. This value must lie in the range $0.0 \leq s \leq 1.0$.

Repeat this data line as often as necessary to define the ε^{ms}–s relationship from $s = 0.0$ to $s = 1.0$ in increasing values of s.

13.20 *MOLECULAR WEIGHT: Define the molecular weight of an ideal gas species.

This option is used to define the molecular weight of an ideal gas species. It can be used only in conjunction with the *FLUID BEHAVIOR option.

Product: Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Fluid cavity definition," Section 11.6.2 of the Abaqus Analysis User's Manual
- "Inflator definition," Section 11.6.4 of the Abaqus Analysis User's Manual
- *FLUID BEHAVIOR
- *FLUID CAVITY

There are no parameters associated with this option.

Data line to define the molecular weight:

First (and only) line:

1. Molecular weight of the ideal gas species.

13.21 *MONITOR: Define a degree of freedom to monitor.

This option is used to choose a node and degree of freedom to monitor the progress of the solution in the status file. In Abaqus/Standard the information will also be written to the message file.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

Reference:

- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual

Required parameters:

DOF

Set this parameter equal to the degree of freedom to be monitored at the node. In an Abaqus/Explicit analysis the degree of freedom will be in the global coordinate system. If the *TRANSFORM option is used at the node in an Abaqus/Standard analysis, the degree of freedom is in the local, transformed, system.

NODE

Set this parameter equal to either the node number to be monitored or the name of a node set containing the node to be monitored. If the name of a node set is chosen, the node set must contain exactly one node.

Optional parameter:

FREQUENCY

This parameter applies only to Abaqus/Standard analyses.

This parameter will only affect output to the message file. Set this parameter equal to the output frequency in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

There are no data lines associated with this option.

13.22 *MOTION: Specify motions as a predefined field.

This option is used to specify motions of node sets or individual nodes during cavity radiation heat transfer analysis, to define the motion of a reference frame in steady-state transport analysis, or to define the velocity of the material transported through the mesh during a static analysis.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual
- "Steady-state transport analysis," Section 6.4.1 of the Abaqus Analysis User's Manual
- "Static stress analysis," Section 6.2.2 of the Abaqus Analysis User's Manual
- "UMOTION," Section 1.1.39 of the Abaqus User Subroutines Reference Manual

Optional, mutually exclusive parameters:

ROTATION

Include this parameter to define a rigid body rotation about an axis.

TRANSLATION

Include this parameter to give the x-, y-, and z-components of translation in the global coordinate system or in the local coordinate system if *TRANSFORM was used at these nodes. Translational motion is the default.

USER

Include this parameter to indicate that magnitudes of motion will be defined in user subroutine **UMOTION**. If this parameter is used, any magnitudes defined by the data lines can be redefined in the user subroutine. The value of the TYPE parameter is not relevant when this parameter is used. This parameter cannot be used for steady-state transport analysis.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve (defined in the *AMPLITUDE option) that gives the time variation of the motion throughout the step ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual).

If this parameter is omitted and the translational or rotational motion is given with TYPE=DISPLACEMENT, the default is a RAMP function. If the translational or rotational motion

is given with TYPE=VELOCITY, the default is a STEP function for cavity radiation analysis and a RAMP function for steady-state transport analysis.

TYPE

This parameter is used to specify whether the magnitude is in the form of a displacement or a velocity.

Set TYPE=DISPLACEMENT (default for cavity radiation analysis) to give translational or rotational displacement values.

Set TYPE=VELOCITY (only type available for steady-state transport and static analysis) to give translational or rotational velocities. Velocity histories for cavity radiation problems can be specified as illustrated in the discussion on prescribing large rotations in "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual.

Data lines to define translational motion (TRANSLATION):

First line:

1. Node set label or node number.
2. First translational component of motion prescribed (only degrees of freedom 1, 2, or 3 can be entered). See "Conventions," Section 1.2.2 of the Abaqus Analysis User's Manual, for a definition of the numbering of degrees of freedom in Abaqus.
3. Last translational component of motion prescribed (only degrees of freedom 1, 2, or 3 can be entered). This field can be left blank if motion for only one component is being prescribed.
4. Magnitude of the translational displacement or velocity. This magnitude will be modified by the *AMPLITUDE specification if the AMPLITUDE parameter is used.

Repeat this data line as often as necessary to define translational motion for different nodes and degrees of freedom.

Data lines to define rotational motion (ROTATION):

First line:

1. Node set label or node number.
2. Magnitude of the rotation (in radians) or rotational velocity (in radians/time). This magnitude will be modified by the *AMPLITUDE specification if the AMPLITUDE parameter is used. The rotation is about the axis defined from point a to point b, where the coordinates of a and b are given next. In steady-state transport analysis the position and orientation of the rotation axis are applied at the beginning of the step and remain fixed during the step.
3. Global x-component of point a on the axis of rotation.
4. Global y-component of point a on the axis of rotation.

The following data are required only for three-dimensional cases:

5. Global z-component of point a on the axis of rotation.
6. Global x-component of point b on the axis of rotation.
7. Global y-component of point b on the axis of rotation.
8. Global z-component of point b on the axis of rotation.

Repeat this data line as often as necessary to define rotational motion for different nodes.

Data lines to define motion in user subroutine `UMOTION` (USER):

First line:

1. Node set label or node number.
2. First translational component of motion prescribed (only degrees of freedom 1, 2, or 3 can be entered). See "Conventions," Section 1.2.2 of the Abaqus Analysis User's Manual, for a definition of the numbering of degrees of freedom in Abaqus.
3. Last translational component of motion prescribed (only degrees of freedom 1, 2, or 3 can be entered). This field can be left blank if motion for only one component is being prescribed.
4. Magnitude of the translational displacement or rotation. This magnitude can be redefined in user subroutine **`UMOTION`**.

Repeat this data line as often as necessary to define the nodes and degrees of freedom that will have their motion prescribed by user subroutine **`UMOTION`**.

13.23 *MPC: Define multi-point constraints.

This option is used to impose constraints between different degrees of freedom of the model.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- "General multi-point constraints," Section 31.2.2 of the Abaqus Analysis User's Manual
- "MPC," Section 1.1.14 of the Abaqus User Subroutines Reference Manual

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

MODE

This parameter applies only to Abaqus/Standard analyses.
This parameter is used only if the USER parameter is included.
Set MODE=DOF (default) for user subroutine **MPC** to operate in a degree of freedom mode.
Set MODE=NODE for user subroutine **MPC** to operate in a nodal mode.

USER

This parameter applies only to Abaqus/Standard analyses.
Include this parameter to indicate that the constraint is defined in user subroutine **MPC**.

Data lines to define multi-point constraints:

First line:

1. MPC type from "General multi-point constraints," Section 31.2.2 of the Abaqus Analysis User's Manual, or, if the USER parameter is included, an integer key to be used in user subroutine **MPC** to distinguish between different constraint types.
2. Node numbers or node sets involved in the constraint.

The first 15 nodes or node sets of an MPC must be entered on the first line. If the MPC contains more than 15 nodes, enter 0 on the next line to indicate that it is a continuation line and then continue to enter

the following nodes on this line. Any number of continuation lines are allowed. Exactly 15 nodes or node sets must be given on each line except the last line.

13.24 *MULLINS EFFECT: Specify Mullins effect material parameters for elastomers.

This option is used to define material constants for the Mullins effect in filled rubber elastomers or for modeling energy dissipation in elastomeric foams. It can be used only with the *HYPERELASTIC or the *HYPERFOAM options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Hyperelastic behavior of rubberlike materials," Section 19.5.1 of the Abaqus Analysis User's Manual
- "Hyperelastic behavior in elastomeric foams," Section 19.5.2 of the Abaqus Analysis User's Manual
- "Mullins effect in rubberlike materials," Section 19.6.1 of the Abaqus Analysis User's Manual
- "Energy dissipation in elastomeric foams," Section 19.6.2 of the Abaqus Analysis User's Manual
- "UMULLINS," Section 1.1.40 of the Abaqus User Subroutines Reference Manual
- *BIAXIAL TEST DATA
- *PLANAR TEST DATA
- *UNIAXIAL TEST DATA

Optional, mutually exclusive parameters:

TEST DATA INPUT

Include this parameter if the material constants are to be computed by Abaqus from data taken from simple tests on a material specimen. If this parameter is omitted in Abaqus/Standard, the material constants may be given directly on the data lines or the damage variable may be defined through user subroutine **UMULLINS**. If this parameter is omitted in Abaqus/Explicit, the material constants must be given directly on the data lines.

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the damage variable defining the Mullins effect is defined in user subroutine **UMULLINS**.

Optional parameters:

BETA

This parameter can be used only when the TEST DATA INPUT parameter is used; it defines the value of β while the other coefficients of the Mullins effect model are fitted from the test data. It cannot be specified if both the R and M parameters are also specified (use the data line instead to specify all three parameters). If this parameter is omitted, β will be determined from a nonlinear, least-squares fit of the test data. Allowable values of BETA are $\beta \geq 0$. The M and BETA parameters cannot both be zero.

DEPENDENCIES

Set this parameter equal to the number of field variables, in addition to temperature, on which the material parameters depend. If this parameter is omitted, it is assumed that the material parameters are constant or depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

This parameter is not relevant if the USER or the TEST DATA INPUT parameter is included.

M

This parameter can be used only when the TEST DATA INPUT parameter is used; it defines the value of m while the other coefficients of the Mullins effect model are fitted from the test data. It cannot be specified if both the R and BETA parameters are also specified (use the data line instead to specify all three parameters). If this parameter is omitted, m will be determined from a nonlinear, least-squares fit of the test data. Allowable values of M are $m \geq 0$. The M and BETA parameters cannot both be zero.

PROPERTIES

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only if the USER parameter is specified. Set this parameter equal to the number of property values needed as data in user subroutine **UMULLINS**. The default value is 0.

R

This parameter can be used only when the TEST DATA INPUT parameter is used; it defines the value of r while the other coefficients of the Mullins effect model are fitted from the test data. It cannot be specified if both the M and BETA parameters are also specified (use the data line instead to specify all three parameters). If this parameter is omitted, r will be determined from a nonlinear, least-squares fit of the test data. Allowable values of R are $r > 1$.

To define the material behavior by giving test data:

No data lines are used with this option when the TEST DATA INPUT parameter is specified. The data are given instead under the *BIAXIAL TEST DATA, *PLANAR TEST DATA, and *UNIAXIAL TEST DATA options. These options are applicable except for the case where the damage variable is defined by the user.

Data lines to define the material constants if both the TEST DATA INPUT and USER parameters are omitted:

First line:

1. r.
2. m.
3. β (If this entry is left blank, the default value is taken to be 0.0 in Abaqus/Standard and 0.1 in Abaqus/Explicit.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the material constants as a function of temperature and other predefined field variables.

Data lines to define the material properties if the USER parameter is specified:

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Material properties, eight per line.

Repeat this data line as often as necessary to define the material properties.

13.25 *M1: Define the first bending moment behavior of beams.

This option is used to define the first bending moment behavior of beams. It can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Using a general beam section to define the section behavior," Section 26.3.7 of the Abaqus Analysis User's Manual
- *BEAM GENERAL SECTION

Optional parameters (if neither ELASTIC nor LINEAR is included, elastic-plastic response is assumed):

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the moment-curvature relationship, in addition to temperature. If this parameter is omitted, it is assumed that the moment–curvature relationship is constant or depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

ELASTIC

Include this parameter if the bending moment–curvature relationship is nonlinear but elastic.

LINEAR

Include this parameter if the bending moment varies linearly with curvature.

Data lines if the LINEAR parameter is included:

First line:

1. Bending stiffness of the section about the first beam section local axis.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the bending stiffness as a function of temperature and other predefined field variables.

Data lines if the LINEAR parameter is omitted:

First line:

1. Bending moment.
2. Curvature.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the bending moment–curvature relationship as a function of temperature and other predefined field variables.

13.26 *M2: Define the second bending moment behavior of beams.

This option is used to define the second bending moment behavior of beams. It can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option and is needed only for beams in space.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Using a general beam section to define the section behavior," Section 26.3.7 of the Abaqus Analysis User's Manual
- *BEAM GENERAL SECTION

Optional parameters (if neither ELASTIC nor LINEAR is included, elastic-plastic response is assumed):

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the moment–curvature relationship, in addition to temperature. If this parameter is omitted, it is assumed that the moment–curvature relationship is constant or depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

ELASTIC

Include this parameter if the bending moment–curvature relationship is nonlinear but elastic.

LINEAR

Include this parameter if the bending moment varies linearly with curvature.

Data lines if the LINEAR parameter is included:

First line:

1. Bending stiffness of the section about the second beam section local axis.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the bending stiffness as a function of temperature and other predefined field variables.

Data lines if the LINEAR parameter is omitted:

First line:

1. Bending moment.
2. Curvature.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the moment–curvature relationship as a function of temperature and other predefined field variables.

14. N

14.1 *NCOPY: Create nodes by copying.

This option is used to copy a node set to create a new node set.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; copying portions of sketches and instancing of parts serve similar purposes.

Reference:

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Manual

Required parameters:

CHANGE NUMBER

Set this parameter equal to an integer that will be added to each of the existing node numbers to define the node numbers of the nodes being created.

OLD SET

Set this parameter equal to the name of the node set being copied. This set will be used for the copy operation with the nodes that belong to it at the time this *NCOPY option appears in the input file.

Required, mutually exclusive parameters:

POLE

Include this parameter if the new nodes are created by projecting the nodes in the old set from the pole node. Each new node will be located such that the corresponding old node is equidistant between the pole node and the new node.

This parameter is particularly useful for creating nodes associated with infinite elements ("Infinite elements," Section 25.3.1 of the Abaqus Analysis User's Manual).

REFLECT

Set REFLECT=LINE to create the new nodes by reflection through a line.

Set REFLECT=MIRROR to create the new nodes by reflection through a plane.

Set REFLECT=POINT to create the new nodes by reflection through a point.

SHIFT

Include this parameter if the new nodes are to be created by translation and/or rotation of the nodes in the old node set. If both translation and rotation are specified, the translation is applied once before the rotation.

Optional parameters:

MULTIPLE

This parameter is used with the SHIFT parameter to define the number of times the rotation should be applied. The default is MULTIPLE=1.

NEW SET

Set this parameter equal to the name of the node set to which the nodes created by the operation will be assigned. This new node set will be unsorted if the OLD SET was unsorted and if the NEW SET does not already exist. Otherwise, this new node set will be a sorted set.

If this parameter is omitted, the newly created nodes are not assigned to a node set.

Data lines if the SHIFT parameter is included:

First line:

1. Value of the translation to be applied in the *X*-direction.
2. Value of the translation to be applied in the *Y*-direction.
3. Value of the translation to be applied in the *Z*-direction.

Second line:

1. *X*-coordinate of the first point defining the rotation axis (point *a* in Figure 14.1–1).
2. *Y*-coordinate of the first point defining the rotation axis.
3. *Z*-coordinate of the first point defining the rotation axis.
4. *X*-coordinate of the second point defining the rotation axis (point *b* in Figure 14.1–1).
5. *Y*-coordinate of the second point defining the rotation axis.
6. *Z*-coordinate of the second point defining the rotation axis.
7. Angle of rotation about the axis *a*–*b*, in degrees.

Data line if REFLECT=LINE:

First (and only) line:

1. *X*-coordinate of the first point defining the reflection line (point *a* in Figure 14.1–2).
2. *Y*-coordinate of the first point defining the reflection line.
3. *Z*-coordinate of the first point defining the reflection line.
4. *X*-coordinate of the second point defining the reflection line (point *b* in Figure 14.1–2).
5. *Y*-coordinate of the second point defining the reflection line.
6. *Z*-coordinate of the second point defining the reflection line.

Data lines if REFLECT=MIRROR:

First line:

1. *X*-coordinate of the first point defining the reflection plane (point *a* in Figure 14.1–3).
2. *Y*-coordinate of the first point defining the reflection plane.

3. *Z*-coordinate of the first point defining the reflection plane.
4. *X*-coordinate of the second point defining the reflection plane (point *b* in Figure 14.1–3).
5. *Y*-coordinate of the second point defining the reflection plane.
6. *Z*-coordinate of the second point defining the reflection plane.

Second line:

1. *X*-coordinate of the third point defining the reflection plane (point *c* in Figure 14.1–3).
2. *Y*-coordinate of the third point defining the reflection plane.
3. *Z*-coordinate of the third point defining the reflection plane.

Data line if REFLECT=POINT:

First (and only) line:

1. *X*-coordinate of the reflection point (point *a* in Figure 14.1–4).
2. *Y*-coordinate of the reflection point.
3. *Z*-coordinate of the reflection point.

Data line if the POLE parameter is included:

First (and only) line:

1. Number of the pole node (optional: it must have been defined already).
2. *X*-coordinate of the pole node (point *a* in Figure 14.1–5, only required if the pole node number was not entered).
3. *Y*-coordinate of the pole node (only required if the pole node number was not entered).
4. *Z*-coordinate of the pole node (only required if the pole node number was not entered).

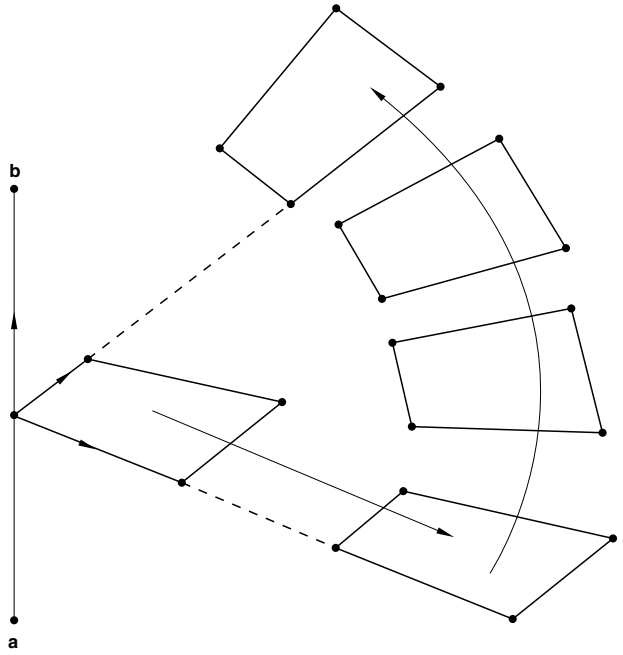


Figure 14.1–1 *NCOPY, SHIFT option.

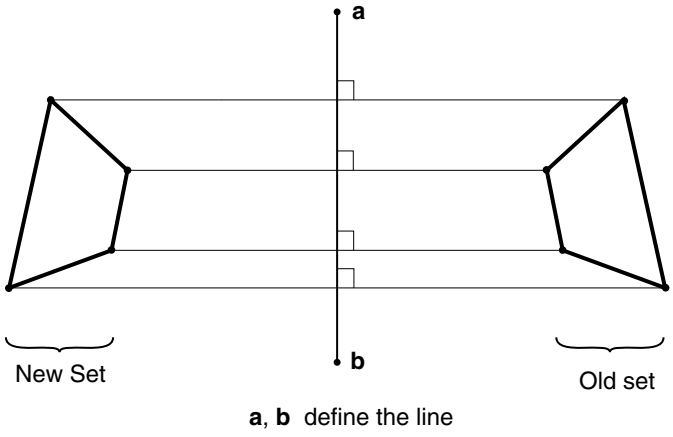


Figure 14.1–2 *NCOPY, REFLECT=LINE option.

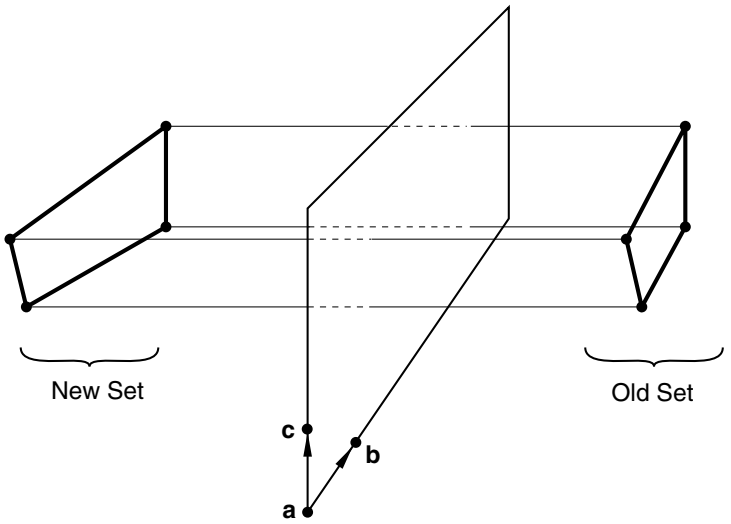


Figure 14.1–3 *NCOPY, REFLECT=MIRROR option.

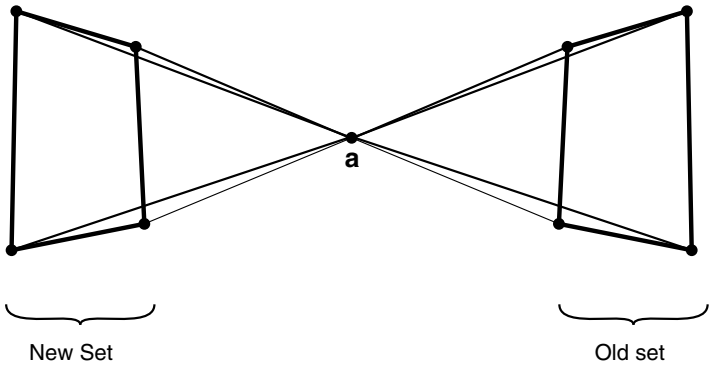


Figure 14.1–4 *NCOPY, REFLECT=POINT option.

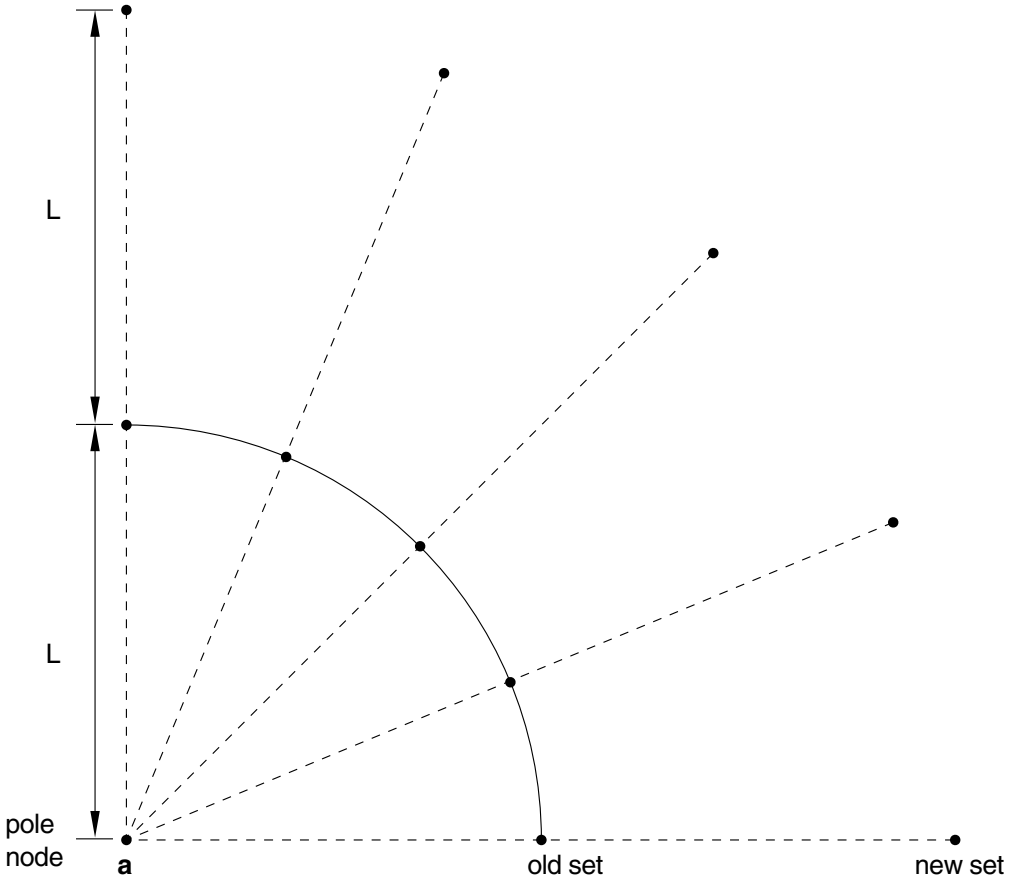


Figure 14.1–5 *NCOPY, POLE option.

14.2 *NFILL: Fill in nodes in a region.

This option is used to generate nodes for a region of a mesh by filling in nodes between two bounds.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; nodes are generated when you mesh the model.

Reference:

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

BIAS

Include this parameter to bias the spacing of the nodes being generated toward one end of the line of nodes being generated. Set this parameter equal to the ratio of adjacent distances between nodes along each line of nodes generated, as the nodes go from the first bounding node set to the second. If the BIAS parameter is less than one, the nodes are concentrated toward the first bounding node set; if it is greater than one, the nodes are concentrated toward the second bounding set. The value of the parameter must be positive.

The BIAS and SINGULAR parameters are mutually exclusive.

NSET

Set this parameter equal to the name of the node set to which the nodes created by this operation (including the bounding nodes) will be assigned. Node sets generated by this option are always sorted node sets.

SINGULAR

This parameter applies only to Abaqus/Standard analyses.

Include this parameter for fracture mechanics calculations with second-order isoparametric elements to create a "quarter point" crack-tip element and bias the remaining elements. Set SINGULAR=1 or 2, depending on whether the first or the second bounding node set represents the crack tip.

The BIAS and SINGULAR parameters are mutually exclusive.

TWO STEP

This parameter is meaningful only if the BIAS parameter is used. When this parameter is included, the BIAS value is applied only at each second interval along the line; therefore, the midside nodes of second-order elements will be at the middle of the two adjacent intervals.

Data lines to fill in nodes between two bounds:

First line:

1. Name of the node set defining the first bound of the region.
2. Name of the node set defining the second bound of the region.
3. Number of intervals along each line between bounding nodes.
4. Increment in node numbers from the node number at the first bound set end. The default is 1.

Repeat this data line as often as necessary, one line per region to be filled by this option.

14.3 *NGEN: Generate incremental nodes.

This option is used to generate nodes incrementally.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; nodes are generated when you mesh the model.

Reference:

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

LINE

Set LINE=P to generate the nodes along a parabola. In this case the user must define an extra point, the midpoint between the two end points.

Set LINE=C to generate the nodes along a circular arc. In this case the user must define an extra point, the center of the circle.

If this parameter is omitted, the nodes will be generated along a straight line.

NSET

Set this parameter equal to the name of a node set to which the nodes will be assigned. The two end nodes will also be included in the node set. Node sets created or modified with this option will always be sorted.

SYSTEM

Set SYSTEM=RC (default) to define the extra node in a Cartesian coordinate system. Set SYSTEM=C to define the extra node in a cylindrical coordinate system. Set SYSTEM=S to define the extra node in a spherical coordinate system. See Figure 14.3–1.

Data lines to generate nodes incrementally:

First line:

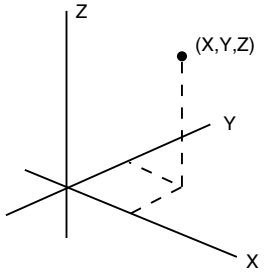
1. Number of the first end node (it must have been previously defined).
2. Number of the second end node (it must have been previously defined).
3. Increment in the numbers between each node along the line. The default is 1.

4. Number of the third node giving the extra point (if required; it must have been previously defined).
5. First coordinate of the extra point (if required).
6. Second coordinate of the extra point (if required).
7. Third coordinate of the extra point (if required).

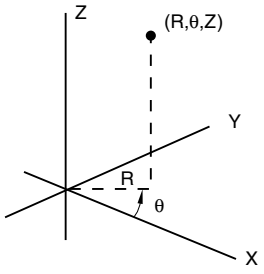
The following entries are used only for a circular arc equal to or larger than 180°:

8. First component of a normal to the circular arc.
9. Second component of a normal to the circular arc.
10. Third component of a normal to the circular arc.

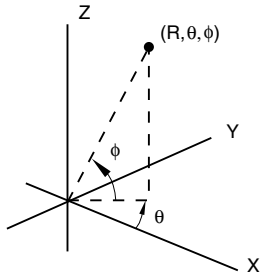
Repeat this data line as often as necessary.



Rectangular Cartesian
(default)



Cylindrical
(θ and ϕ are given in degrees)



Spherical

Figure 14.3–1 Coordinate systems.

14.4 *NMAP: Map nodes from one coordinate system to another and rotate, translate, or scale the nodal coordinates.

Map nodes from one coordinate system to another and rotate, translate, or scale the nodal coordinates.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Abaqus/CAE: Unsupported; meshing techniques in the Mesh module are usually preferable.

Reference:

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Manual

Required parameters:

NSET

Set this parameter equal to the name of the node set containing the nodes to be mapped. The nodes that are mapped are those that belong to this set at the time this option is encountered.

TYPE

Set TYPE=ROTATION to introduce a rotation of a specified angle about a given axis defined by two points *a* and *b* (or by the coordinates of these points). The origin of rotation is given by a third point *c* (or by the coordinates of this point).

Set TYPE=TRANSLATION to introduce a translation along a given axis defined by two nodes *a* and *b* (or by the coordinates of these points).

Set TYPE=SCALE to scale each axis with respect to one node *a* (or by the coordinates of this point).

Set TYPE=RECTANGULAR to introduce a simple shift or rotation. Point *a* in Figure 14.4–1 defines the origin of the local rectangular coordinate system defining the map. The local $\hat{x}$-axis is defined by the line joining points *a* and *b*. The local $\hat{x}$–$\hat{y}$ plane is defined by the plane passing through points *a*, *b*, and *c*.

Set TYPE=CYLINDRICAL to map from cylindrical coordinates. Point *a* in Figure 14.4–1 defines the origin of the local cylindrical coordinate system defining the map. The line going through point *a* and point *b* defines the $\hat{z}$-axis of the local cylindrical coordinate system. The local $\hat{r}$–$\hat{z}$ plane for $\theta = 0$ is defined by the plane passing through points *a*, *b*, and *c*.

Set TYPE=DIAMOND to map from skewed Cartesian coordinates. Point *a* in Figure 14.4–1 defines the origin of the local diamond coordinate system defining the map. The line going through point *a* and point *b* defines the $\hat{x}$-axis of the local coordinate system. The line going through point

a and point c defines the $\hat{y}$-axis of the local coordinate system. The line going through point a and point d defines the $\hat{z}$-axis of the local coordinate system.

Set TYPE=SPHERICAL to map from spherical coordinates. Point a in Figure 14.4–1 defines the origin of the local spherical coordinate system defining the map. The line going through point a and point b defines the polar axis of the local spherical coordinate system. The plane passing through point a and perpendicular to the polar axis defines the $\phi = 0$ plane. The plane passing through points a, b, and c defines the local $\theta = 0$ plane.

Set TYPE=TOROIDAL to map from toroidal coordinates. Point a in Figure 14.4–1 defines the origin of the local toroidal coordinate system defining the map. The axis of the local toroidal system lies in the plane defined by points a, b, and c. The R-coordinate of the toroidal system is defined by the distance between points a and b. The line between points a and b defines the $\phi = 0$ position. For every value of ϕ the θ-coordinate is defined in a plane perpendicular to the plane defined by the points a, b, and c and perpendicular to the axis of the toroidal system. $\theta = 0$ lies in the plane defined by the points by a, b, and c.

Set TYPE=BLENDED to map via blended quadratics in an Abaqus/Standard analysis.

Optional parameter:

DEFINITION

Set DEFINITION =COORDINATES (default) to define the local system, the axis of rotation, the origin of rotation, or the axis of translation by giving the coordinates of the points a, b, c, and d whichever appropriate for the chosen type.

Set DEFINITION=NODES to define the local system, the axis of rotation, the origin of rotation, or the axis of translation by giving global node numbers for points a, b, c, and d depending on the type. This option cannot be used with TYPE=BLENDED.

Data lines for TYPE=ROTATION, DEFINITION=COORDINATES:

First line:

1. X-coordinate of point a.
2. Y-coordinate of point a.
3. Z-coordinate of point a.
4. X-coordinate of point b.
5. Y-coordinate of point b.
6. Z-coordinate of point b.

Second line:

1. X-coordinate of point c.
2. Y-coordinate of point c.
3. Z-coordinate of point c.

Third line:

1. The rotation angle in degrees.

Data lines for TYPE=ROTATION, DEFINITION=NODES:

First line:

1. Local node number of point *a*.
2. Local node number of point *b*.

Second line:

1. Local node number of point *c*.

Third line:

1. The rotation angle in degrees.

Data lines for TYPE=TRANSLATION, DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. The translation magnitude.

Data lines for TYPE=TRANSLATION, DEFINITION=NODES:

First line:

1. Local node number of point *a*.
2. Local node number of point *b*.

Second line:

1. The translation magnitude.

Data lines for TYPE=SCALE, DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a*.

2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.

Second line:

1. Scale factor to be applied to the first local coordinate.
2. Scale factor to be applied to the second local coordinate.
3. Scale factor to be applied to the third local coordinate.

Data lines for TYPE=SCALE, DEFINITION=NODES:

First line:

1. Local node number of point *a*.

Second line:

1. Scale factor to be applied to the first local coordinate before mapping.
2. Scale factor to be applied to the second local coordinate before mapping.
3. Scale factor to be applied to the third local coordinate before mapping.

Data lines for TYPE=RECTANGULAR, CYLINDRICAL, DIAMOND, SPHERICAL, or TOROIDAL with DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a* (see Figure 14.4–1).
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. *X*-coordinate of point *c*.
2. *Y*-coordinate of point *c*.
3. *Z*-coordinate of point *c*.

The following fields are needed only for TYPE=DIAMOND:

4. *X*-coordinate of point *d*.
5. *Y*-coordinate of point *d*.
6. *Z*-coordinate of point *d*.

If TYPE=RECTANGULAR is specified and only point *a* is given, the coordinates of the nodes in the set are simply shifted by X_a, Y_a, and Z_a.

Third line:

1. Scale factor to be applied to the first local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.
2. Scale factor to be applied to the second local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.
3. Scale factor to be applied to the third local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.

Data lines for TYPE=RECTANGULAR, CYLINDRICAL, DIAMOND, SPHERICAL, or TOROIDAL with DEFINITION=NODES:

First line:

1. Local node number of point *a*.
2. Local node number of point *b*.

Second line:

1. Local node number of point *c*.

The following field is needed only for TYPE=DIAMOND:

2. Local node number of point *d*.

If TYPE=RECTANGULAR is specified and only point *a* is given, the coordinates of the nodes in the set are simply shifted by X_a, Y_a, and Z_a.

Third line:

1. Scale factor to be applied to the first local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.
2. Scale factor to be applied to the second local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.
3. Scale factor to be applied to the third local coordinate before mapping. If the value entered is zero or blank, a scale factor of 1.0 is assumed.

Data lines for TYPE=BLENDED:

First line:

1. Node number of the first control node.
2. *X*-coordinate of the point to which this control node is to be mapped.
3. *Y*-coordinate of the point to which this control node is to be mapped.
4. *Z*-coordinate of the point to which this control node is to be mapped.

Second line:

1. Node number of the second control node.

2. *X*-coordinate of the point to which this control node is to be mapped.
3. *Y*-coordinate of the point to which this control node is to be mapped.
4. *Z*-coordinate of the point to which this control node is to be mapped.

Continue, giving up to 20 control nodes, but giving at least the eight corner nodes. If an edge of the blended mapping is to be mapped linearly, the corresponding mid-edge control node can be omitted from the list. This is done by inserting a line with node number 0 only (a blank line) in place of the definition of the control node and its mapped coordinates. The control nodes do not have to be nodes in the finite element model—they can be nodes used just for mesh generation. Abaqus eliminates any nodes that are not used in the analysis model before doing the analysis.

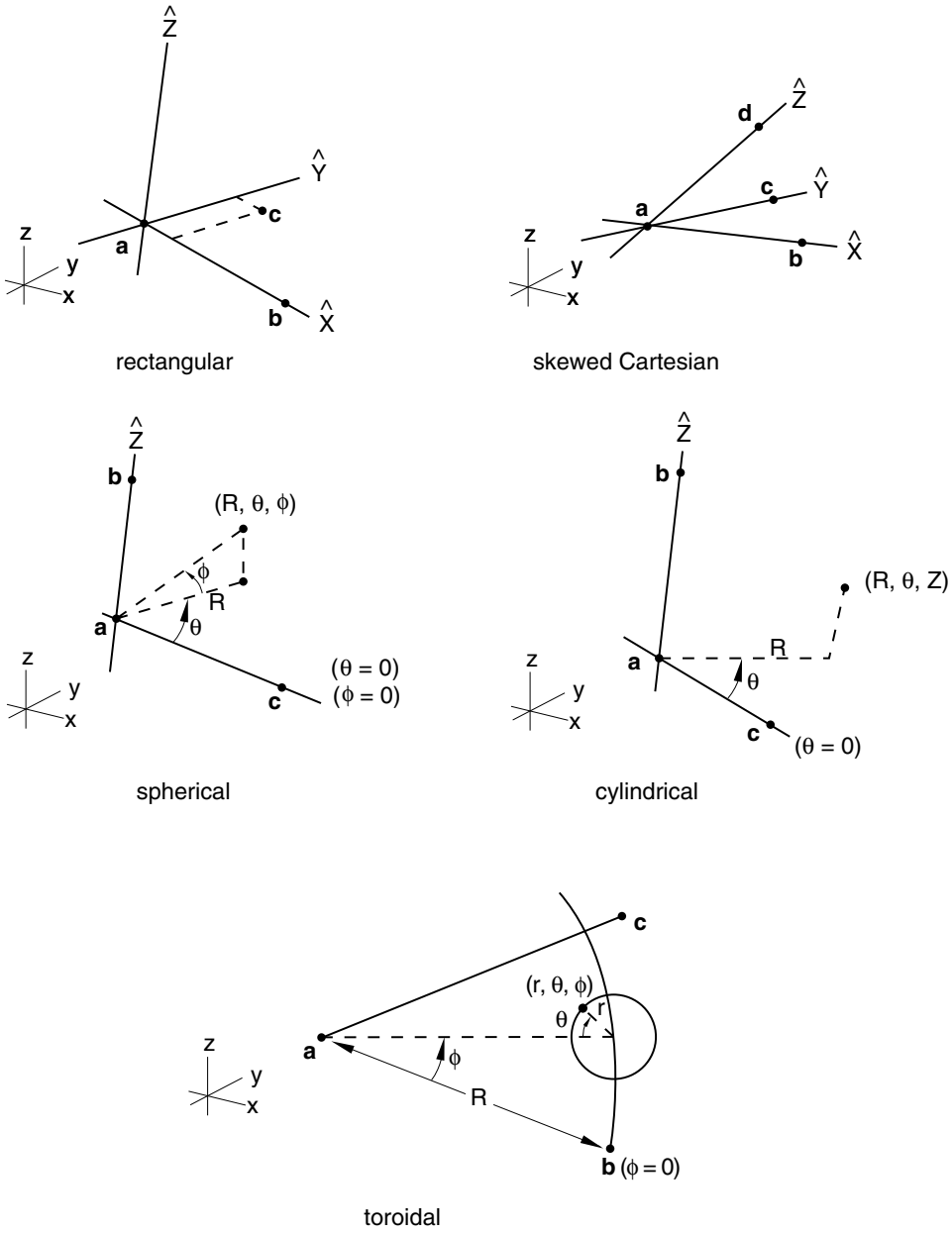


Figure 14.4–1 Coordinate systems; angles are in degrees.

14.5 *NO COMPRESSION: Introduce a compressive failure theory (tension only materials).

This option is used to modify the elasticity definition so that no compressive stress is allowed. It can be used only in conjunction with the *ELASTIC option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "No compression or no tension," Section 19.2.2 of the Abaqus Analysis User's Manual
- *ELASTIC

There are no parameters or data lines associated with this option.

14.6 *NO TENSION: Introduce a tension failure theory (compression only material).

This option is used to modify the elasticity definition so that no tensile stress is allowed. It can be used only in conjunction with the *ELASTIC option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "No compression or no tension," Section 19.2.2 of the Abaqus Analysis User's Manual
- *ELASTIC

There are no parameters or data lines associated with this option.

14.7 *NODAL ENERGY RATE: Define critical energy release rates at nodes.

This option is used to define variable critical energy release rates on a nodal basis. The critical energy release rate data defined with this option is ignored unless the NODAL ENERGY RATE parameter is included on the *FRACTURE CRITERION, TYPE=VCCT option or the *FRACTURE CRITERION, TYPE=FATIGUE option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Crack propagation analysis," Section 11.4.3 of the Abaqus Analysis User's Manual
- *FRACTURE CRITERION

Optional parameters:

GENERATE

Include this parameter to interpolate the critical energy release rates between two bounding nodes or node sets. The critical energy release rates for the bounding nodes or node sets must have been defined earlier. If the node sets do not have the same number of nodes, the extra nodes in the longer set are ignored.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines when the GENERATE parameter is omitted:

First line:

1. Node set label or node number.
2. G_{IC}, Mode I critical energy release rate.
3. G_{IIC}, Mode II critical energy release rate.
4. G_{IIIC}, Mode III critical energy release rate.

Repeat this data line as often as necessary to define the variation in critical energy release rates.

Data lines when the GENERATE parameter is included:

First line:

1. Node number or node set label that defines the first bound for the generate operation.
2. Node number or node set label that defines the second bound for the generate operation.
3. Number of intervals between the bounding nodes or node sets.
4. Increment in node numbers from the first bounding node or node set.

Repeat this data line as often as necessary to define the variation in critical energy release rates.

14.8 *NODAL THICKNESS: Define shell or membrane thickness at nodes.

This option is used to define variable shell or membrane thicknesses on a nodal basis. The thickness data defined with this option will be ignored unless the NODAL THICKNESS parameter is included on either the *SHELL GENERAL SECTION or the *SHELL SECTION options for shell elements or on the *MEMBRANE SECTION option for membrane elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- "Nodal thicknesses," Section 2.1.3 of the Abaqus Analysis User's Manual
- "Membrane elements," Section 26.1.1 of the Abaqus Analysis User's Manual
- "Using a shell section integrated during the analysis to define the section behavior," Section 26.6.5 of the Abaqus Analysis User's Manual
- "Using a general shell section to define the section behavior," Section 26.6.6 of the Abaqus Analysis User's Manual
- "Line spring elements for modeling part-through cracks in shells," Section 29.10.1 of the Abaqus Analysis User's Manual
- *MEMBRANE SECTION
- *SHELL GENERAL SECTION
- *SHELL SECTION

Optional parameters:

GENERATE

Include this parameter to interpolate the thickness between two bounding nodes or node sets. The thickness for the bounding nodes or node sets must have been defined earlier. If the node sets do not have the same number of nodes, the extra nodes in the longer set are ignored.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines when the GENERATE parameter is omitted:

First line:

1. Node set label or node number.
2. Thickness.

Repeat this data line as often as necessary to define the variation in shell or membrane thickness.

Data lines when the GENERATE parameter is included:

First line:

1. Node number or node set label that defines the first bound for the generate operation.
2. Node number or node set label that defines the second bound for the generate operation.
3. Number of intervals between the bounding nodes or node sets.
4. Increment in node numbers from the first bounding node or node set.

Repeat this data line as often as necessary to define the variation in thickness.

14.9 *NODE: Specify nodal coordinates.

This option is used to define a node directly by specifying its coordinates. Nodal coordinates given in this option are in a local system if the *SYSTEM option is in effect when this option is used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Mesh module

Reference:

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

NSET

Set this parameter equal to the name of the node set to which these nodes will be assigned. Node sets created or modified with this option will always be sorted.

SYSTEM

Set SYSTEM=R (default) to give coordinates in a rectangular Cartesian coordinate system. Set SYSTEM=C to give coordinates in a cylindrical system. Set SYSTEM=S to give coordinates in a spherical system. See Figure 14.9–1.

The SYSTEM parameter is entirely local to this option. As the data lines are read, the coordinates given are transformed to rectangular Cartesian coordinates immediately. If the *SYSTEM option is also in effect, these are local rectangular Cartesian coordinates, which are then immediately transformed to global Cartesian coordinates.

Data lines to define the node:

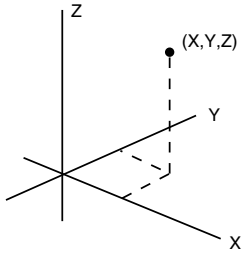
First line:

1. Node number.
2. First coordinate of the node.
3. Second coordinate of the node.

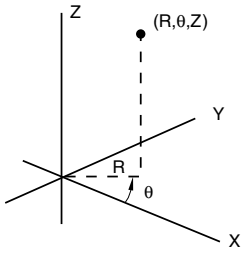
4. Third coordinate of the node.
5. First direction cosine of the normal at the node (optional).
6. Second direction cosine of the normal at the node (optional). For nodes entered in a cylindrical or spherical system, this entry is an angle given in degrees.
7. Third direction cosine of the normal at the node (optional). For nodes entered in a spherical system, this entry is an angle given in degrees.

The normal will be used only for element types with rotational degrees of freedom. See Part VI, "Elements," of the Abaqus Analysis User's Manual.

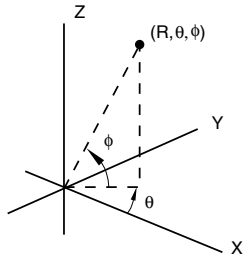
Repeat this data line as often as necessary.



Rectangular Cartesian
(default)



Cylindrical
(θ and ϕ are given in degrees)



Spherical

Figure 14.9–1 Coordinate systems.

14.10 *NODE FILE: Define results file requests for nodal data.

This option is used to choose the nodal variables that will be written to the results (`.fil`) file in an Abaqus/Standard analysis or to the selected results (`.sel`) file in an Abaqus/Explicit analysis. In an Abaqus/Explicit analysis it must be used in conjunction with the *FILE OUTPUT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Manual
- *FILE OUTPUT

Optional parameters:

FREQUENCY

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the output frequency, in increments. The output will always be written to the results file at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

GLOBAL

This parameter applies only to Abaqus/Standard analyses.

This parameter is relevant only at nodes where the *TRANSFORM option has been used to define a local coordinate system.

Set GLOBAL=NO to write vector-valued nodal variables in the local directions.

Set GLOBAL=YES (default) to write vector-valued nodal variables in the global directions. This default is the opposite of the default on the *NODE PRINT option and is used because most postprocessors assume that components are given in the global system.

LAST MODE

This parameter applies only to Abaqus/Standard analyses.

This parameter is useful only during eigenvalue extraction for natural frequencies ("Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Manual) and for eigenvalue buckling estimation ("Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Manual).

Set this parameter equal to the highest mode number for which output is required.

The default value is LAST MODE=*N*, where *N* is the number of modes extracted. If the MODE parameter is used, the default value is LAST MODE=*M*, where *M* is the value of the MODE parameter.

MODE

This parameter applies only to Abaqus/Standard analyses.

This parameter is useful only during eigenvalue extraction for natural frequencies and for eigenvalue buckling estimation. Set this parameter equal to the first mode number for which output is required. The default is MODE=1. See also the LAST MODE parameter. When performing a *FREQUENCY analysis, the normalization will follow the format set by the NORMALIZATION parameter. Otherwise, the normalization is such that the largest displacement component in the mode has a magnitude of 1.0.

NSET

Set this parameter equal to the name of the node set for which the output is being written to the results file. If this parameter is omitted, the output will be written for all nodes in the model.

Data lines to request nodal output in the results or selected results file:

First line:

1. Give the identifying keys for the variables to be written to the results or selected results file. The keys are defined in the "Nodal variables" section of "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual, and "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary to define the nodal variables to be written to the results or selected results file.

14.11 *NODE OUTPUT: Define output database requests for nodal data.

This option is used to write nodal variables to the output database. It must be used in conjunction with the *OUTPUT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *OUTPUT

One of the following mutually exclusive parameters is required when the *NODE OUTPUT option is used in conjunction with the *OUTPUT, HISTORY option:

NSET

Set this parameter equal to the name of the node set for which this output request is being made.

TRACER SET

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the name of the tracer set for which this output request is being made.

Optional parameters when the *NODE OUTPUT option is used in conjunction with the *OUTPUT, FIELD option:

NSET

Set this parameter equal to the name of the node set for which this output request is being made.

If this parameter is omitted, the output will be written for all the nodes in the model.

TRACER SET

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the name of the tracer set for which this output request is being made.

This parameter is valid only for displacement output requests.

Optional parameter when the *NODE OUTPUT option is used in conjunction with the *OUTPUT, HISTORY option:

GLOBAL

This parameter is relevant only at nodes where the *TRANSFORM option has been used to define a local coordinate system.

Set GLOBAL=NO to write vector-valued nodal variables in the local directions.

Set GLOBAL=YES (default) to write vector-valued nodal variables in the global directions. This default is the opposite of the default on the *NODE PRINT option and is used because most postprocessors assume that components are given in the global system.

Optional parameter:

VARIABLE

Set VARIABLE=ALL to indicate that all nodal variables applicable to this procedure and material type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default nodal output variables for the current procedure type should be written to the output database. Additional output variables can be requested on the data lines.

If this parameter is omitted, the nodal variables requested for output must be specified on the data lines.

Data lines to request nodal output:

First line:

1. Specify the identifying keys for the variables to be written to the output database. The keys are defined in "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual, and "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary to define the nodal variables to be written to the output database.

14.12 *NODE PRINT: Define print requests for nodal variables.

This option is used to provide tabular printed output of nodal variables (displacements, reaction forces, etc.) in the data file.

Product: Abaqus/Standard

Type: History data

Level: Step

Reference:

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Manual

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

GLOBAL

This parameter is relevant only at nodes where the *TRANSFORM option has been used to define a local coordinate system.

Set GLOBAL=NO (default) to obtain printout of vector-valued nodal variables in the local directions.

Set GLOBAL=YES to obtain printout of vector-valued nodal variables in the global directions.

LAST MODE

This parameter is useful only during eigenvalue extraction for natural frequencies ("Natural frequency extraction," Section 6.3.5 of the Abaqus Analysis User's Manual), complex eigenvalue extraction ("Complex eigenvalue extraction," Section 6.3.6 of the Abaqus Analysis User's Manual), and eigenvalue buckling estimation ("Eigenvalue buckling prediction," Section 6.2.3 of the Abaqus Analysis User's Manual). Set this parameter equal to the highest mode number for which output is required.

The default value is LAST MODE=*N*, where *N* is the number of modes extracted. If the MODE parameter is used, the default value is LAST MODE=*M*, where *M* is the value of the MODE parameter.

MODE

This parameter is useful only during natural frequency extraction, complex eigenvalue extraction, and eigenvalue buckling estimation. Set this parameter equal to the first mode number for which

output is required. The default is MODE=1. See also the LAST MODE parameter. When performing a *FREQUENCY analysis, the normalization will follow the format set by the NORMALIZATION parameter. Otherwise, the normalization is such that the largest displacement component in the mode has a magnitude of 1.0.

NSET

Set this parameter equal to the name of the node set for which this output request is being made. If this parameter is omitted, the output will be printed for all of the nodes in the model.

SUMMARY

Set SUMMARY=YES (default) to obtain a summary and the locations of the maximum and minimum values in each column of the table. Set SUMMARY=NO to suppress this summary.

TOTALS

Set TOTALS=YES to print the total of each column in the table. This is useful, for example, to sum reaction forces in a particular direction. The default is TOTALS=NO.

Data lines to request nodal output in the data file:

First line:

1. Give the identifying keys for the variables to be printed in a table for this node set. The keys are defined in the "Nodal variables" portion of "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary: each line defines a table. If this line is omitted, no nodal output will be printed to the data file.

14.13 *NODE RESPONSE: Define nodal responses for design sensitivity analysis.

This option is used to write nodal response sensitivities to the output database. It must be used in conjunction with the *DESIGN RESPONSE option.

Product: Abaqus/Design

Type: History data

Level: Step

References:

- "Design sensitivity analysis," Section 16.1.1 of the Abaqus Analysis User's Manual
- *DESIGN RESPONSE

Optional parameter:

NSET

Set this parameter equal to the name of the node set for which this sensitivity output is being made.

Data lines to request nodal sensitivity output:

First line:

1. Specify the identifying keys for the responses whose sensitivities are to be written to the output database. The valid keys are listed in "Design sensitivity analysis," Section 16.1.1 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary to define the nodal responses whose sensitivities are to be written to the output database.

14.14 *NONSTRUCTURAL MASS: Specify mass contribution to the model from nonstructural features.

This option is used to include the mass contribution from nonstructural features in the model. The nonstructural mass can be applied over an element set that contains solid, shell, membrane, surface, beam, or truss elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Property module and Interaction module.

Reference:

- "Nonstructural mass definition," Section 2.6.1 of the Abaqus Analysis User's Manual

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the elements over which a given nonstructural mass is to be distributed.

UNITS

Set UNITS=TOTAL MASS to specify the nonstructural mass in the units of "mass."

Set UNITS=MASS PER VOLUME to specify the nonstructural mass in the units of "mass per unit volume."

Set UNITS=MASS PER AREA to specify the nonstructural mass in the units of "mass per unit area." This value is valid only for an element set that contains conventional shells, membranes, and/or surface elements.

Set UNITS=MASS PER LENGTH to specify the nonstructural mass in the units of "mass per unit length." This value is valid only for an element set that contains beam and/or truss elements.

Optional parameter:

DISTRIBUTION

This parameter is relevant only when UNITS=TOTAL MASS.

Set DISTRIBUTION=MASS PROPORTIONAL (default) to distribute the total nonstructural mass among the members of the element set region in proportion to the element structural mass.

The underlying structural density over the element set region is scaled uniformly; therefore, the center of mass for the element set region is not altered.

Set DISTRIBUTION=VOLUME PROPORTIONAL to distribute the total nonstructural mass among the members of the element set region in proportion to the element volume in the initial configuration. A uniform value is added to the underlying structural density over the element set region; therefore, if the region has nonuniform structural density, the center of mass for the element set region may be altered.

Data line for UNITS=TOTAL MASS:

First (and only) line:

1. Mass magnitude of the nonstructural feature for distribution over the element set region. Mass, not weight, should be given.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

Data line for UNITS=MASS PER VOLUME:

First (and only) line:

1. Mass per unit volume of the nonstructural feature for application over the element set region.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

Data line for UNITS=MASS PER AREA:

First (and only) line:

1. Mass per unit area of the nonstructural feature for application over the element set region.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

Data line for UNITS=MASS PER LENGTH:

First (and only) line:

1. Mass per unit length of the nonstructural feature for application over the element set region.

Abaqus does not use any specific physical units, so the user's choice must be consistent.

14.15 *NORMAL: Specify a particular normal direction.

This option is used to define alternative nodal normals for elements. In an Abaqus/Standard analysis it can also be used to define alternative normals for contact surfaces.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly

Reference:

- "Normal definitions at nodes," Section 2.1.4 of the Abaqus Analysis User's Manual

Optional parameter:

TYPE

Set TYPE=ELEMENT (default) to allow the alternative normal definition of elements.
Set TYPE=CONTACT SURFACE to allow the alternative normal definition of contact surfaces in an Abaqus/Standard analysis.

Data lines to define normals for elements (TYPE=ELEMENT):

First line:

1. Element number or element set label.
2. Node number or node set label.
3. First component of normal in global coordinates.
4. Second component of normal in global coordinates.
5. Third component of normal in global coordinates.

Repeat this data line as often as necessary to define the normals.

Data lines to define normals for contact surfaces (TYPE=CONTACT SURFACE):

First line:

1. Master surface name.
2. Node number or node set label (on master surface).
3. First component of normal in global coordinates.
4. Second component of normal in global coordinates.
5. Third component of normal in global coordinates.

Repeat this data line as often as necessary to define the normals.

14.16 *NSET: Assign nodes to a node set.

This option assigns nodes to a node set.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Part, Part instance, Assembly, Model, Step

Abaqus/CAE: Set toolset

Reference:

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Manual

Required parameter:

NSET

Set this parameter equal to the name of the node set to which the nodes will be assigned.

Optional parameters:

ELSET

Set this parameter equal to the name of a previously defined element set. The nodes that define the elements that belong to this element set at the time this option is encountered will be assigned to the node set specified. The UNSORTED parameter cannot be used with this parameter.

The ELSET and GENERATE parameters are mutually exclusive.

GENERATE

If this parameter is included, each data line should give a first node, n_1; a last node; n_2; and the increment in node numbers between these nodes, i. Then, all nodes going from n_1 to n_2 in steps of i will be added to the set. i must be an integer such that $(n_2 - n_1)/i$ is a whole number (not a fraction).

The ELSET and GENERATE parameters are mutually exclusive.

INSTANCE

Set this parameter equal to the name of the part instance that contains the nodes listed on the data line. This parameter can be used only at the assembly level and is intended to be used as a shortcut to the naming convention. It can be used only in a model defined in terms of an assembly of part instances.

INTERNAL

Abaqus/CAE uses the INTERNAL parameter to identify sets that are created internally. The INTERNAL parameter is used only in models defined in terms of an assembly of part instances. The default is to omit the INTERNAL parameter.

UNSORTED

If this parameter is included, the nodes in this node set will be assigned to the set (or added to the set if it already exists) in the order in which they are given. This parameter will be ignored if the ELSET parameter is used.

If this parameter is omitted, the nodes in the set are sorted into ascending order of their node numbers, with duplicates eliminated.

Data lines if the GENERATE parameter is omitted:

First line:

1. List of nodes or node set labels to be assigned to this node set. Only previously defined node sets can be assigned to another node set.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines if the GENERATE parameter is included:

First line:

1. First node in the set.
2. Last node in the set.
3. Increment in node numbers between nodes in the set. The default is 1.

Repeat this data line as often as necessary.

There are no data lines when the ELSET parameter is specified.

O

15. O

15.1 *ORIENTATION: Define a local axis system for material or element property definition, for kinematic coupling constraints, for free directions for inertia relief loads, or for connectors.

This option is used to define a local coordinate system for definition of material properties; for material calculations at integration points; for element property definitions (e.g., connector elements); for output of components of stress, strain, and element section forces; and for kinematic and distributing coupling constraints. A spatially varying local coordinate system can be defined for solid continuum elements and shell elements using distributions ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual).

The *ORIENTATION option can also be used to specify local material directions for anisotropic hyperelastic materials with invariant-based formulation ("Invariant-based formulation" in "Anisotropic hyperelastic behavior," Section 19.5.3 of the Abaqus Analysis User's Manual). The local directions are defined with respect to the local coordinate system.

In Abaqus/Standard the *ORIENTATION option can be used to define local directions for contact pair interaction properties and spring, dashpot, and JOINTC elements; for definition of local free directions for inertia relief loads; and for output of components of surface variables.

In Abaqus/Explicit the *ORIENTATION option can be used to initialize the directions of the fill and the warp yarns of a fabric material in the reference configuration. The yarn directions lying in the plane of the fabric are defined with respect to the two in-plane axes of the orthogonal coordinate system.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Property module, Interaction module, and Load module

References:

- "Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual
- "ORIENT," Section 1.1.15 of the Abaqus User Subroutines Reference Manual

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the orientation definition. Orientation names in the same input file must be unique.

Optional parameters:

DEFINITION

Set DEFINITION=COORDINATES (default) to define the local system by giving the coordinates of the three points *a*, *b*, and, optionally, *c* (the origin) appropriate to the SYSTEM choice from Figure 15.1–1. A spatially varying local coordinate system can be created for solid continuum elements and shell elements for this parameter value by using a distribution to define spatially varying coordinates for points *a* and *b*. Using a distribution to define the coordinates of the optional point *c* is not currently supported. See "Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual.

Set DEFINITION=NODES to define the local system by giving global node numbers for points *a*, *b*, and, optionally, *c* (the origin).

Set DEFINITION=OFFSET TO NODES to define the local system by giving local node numbers (on the element where the orientation is being used) to define the points *a*, *b*, and, optionally, *c* (the origin) in Figure 15.1–1. This parameter value cannot be used with spring, dashpot, or JOINTC elements. In addition, it cannot be used with the *KINEMATIC COUPLING, *INERTIA RELIEF, or *CONTACT PAIR options.

For all DEFINITION parameter values a spatially varying local coordinate system can be created for solid continuum elements and shell elements by using a distribution to define a spatially varying additional rotation angle α. See "Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual.

LOCAL DIRECTIONS

This parameter is relevant only for anisotropic materials with preferred material directions (or fiber directions), such as anisotropic hyperelastic materials or, in Abaqus/Explicit, fabric materials.

Set this parameter equal to the number of local directions that are applicable to the material model (for example, two for a fabric material). The local directions are specified with respect to the orthonormal system at the material point resulting from the current orientation definition. Up to three local directions can be specified as part of the definition of a local orientation system.

For the fabric material in Abaqus/Explicit, the two yarn directions are given with respect to the in-plane axes of the orthonormal system. If no local directions are specified as part of the orientation definition, the local material directions are assumed to match the in-plane axes of the orthonormal system in the reference configuration.

SYSTEM

Set SYSTEM=RECTANGULAR (default) to define a rectangular Cartesian system by the three points *a*, *b*, and *c* shown in Figure 15.1–1. Point *c* is the origin of the system, point *a* must lie on the X'-axis, and point *b* must lie on the X'-Y' plane. Although not necessary, it is intuitive to select point *b* such that it is on or near the local Y'-axis.

Set SYSTEM=CYLINDRICAL to define a cylindrical system by giving the two points *a* and *b* on the polar axis of the cylindrical system (Figure 15.1–1). The local axes are 1=radial, 2=circumferential, 3=axial.

Set SYSTEM=SPHERICAL to define a spherical system by giving the center of the sphere, *a*, and point *b* on the polar axis (Figure 15.1–1). The local axes are 1=radial, 2=circumferential, 3=meridional.

Set SYSTEM=Z RECTANGULAR to define a rectangular Cartesian system by the three points *a*, *b*, and *c* shown in Figure 15.1–1. Point *c* is the origin of the system, point *a* must lie on the Z'-axis, and point *b* must lie on the X'-Z' plane. Although not necessary, it is intuitive to select point *b* such that it is on or near the local X'-axis.

Set SYSTEM=USER in an Abaqus/Standard analysis to define the local coordinate system in user subroutine **ORIENT**. The DEFINITION parameter and any data lines associated with the option are ignored if SYSTEM=USER.

Data lines to define an orientation using DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

The following items, the coordinates of point *c* (the origin), are optional and relevant only for SYSTEM=RECTANGULAR and SYSTEM=Z RECTANGULAR. The default location of the origin, *c*, is the global origin.

7. *X*-coordinate of point *c*.
8. *Y*-coordinate of point *c*.
9. *Z*-coordinate of point *c*.

Second line:

1. Local direction about which the additional rotation or rotations are given. The default is the local 1-direction. For shell, membrane, and cohesive elements this direction should have a nonzero component in the direction of the normal to the surface.
2. Additional rotation α defined by either a single scalar value or by a distribution. An orientation defined with a distribution can be used only for solid continuum elements and shell elements. The additional rotation (in degrees) is applied to both directions orthogonal to the specified local direction. The default is zero degrees.

Third line when the LOCAL DIRECTIONS parameter is included:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.
2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.

3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Data lines to define a spatially varying orientation for solid continuum and shell elements using a distribution when DEFINITION=COORDINATES:

First line:

1. Distribution name. Data in the distribution define spatially varying coordinates for points *a* and *b*.

Second line:

1. Local direction about which the additional rotation or rotations are given. The default is the local 1-direction. For shell, membrane, and cohesive elements this direction should have a nonzero component in the direction of the normal to the surface.
2. Additional rotation α defined by either a single scalar value or by a distribution. A local coordinate system defined with a distribution can be used only for solid continuum elements and shell elements. The additional rotation (in degrees) is applied to both directions orthogonal to the specified local direction. The default is zero degrees.

Third line when the LOCAL DIRECTIONS parameter is included:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.
2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.
3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Data lines to define an orientation using DEFINITION=NODES:

First line:

1. Node number of the node at point *a*.
2. Node number of the node at point *b*.

The next item, specification of point *c* (the origin), is optional and relevant only for SYSTEM=RECTANGULAR and SYSTEM=Z RECTANGULAR. The default location of the origin, *c*, is the global origin.

3. Node number of the node at point *c*.

Second line:

1. Local direction about which the additional rotation or rotations are given. The default is the local 1-direction. For shell, membrane, and cohesive elements this direction should have a nonzero component in the direction of the normal to the surface.
2. Additional rotation α defined by either a single scalar value or by a distribution. A local coordinate system defined with a distribution can be used only for solid continuum elements and shell elements. The additional rotation (in degrees) is applied to both directions orthogonal to the specified local direction. The default is zero degrees.

Third line when the LOCAL DIRECTIONS parameter is included:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.
2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.
3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

Data lines to define an orientation using DEFINITION=OFFSET TO NODES:

First line:

1. Local node number of point *a*.
2. Local node number of point *b*.

The next item, specification of point *c* (the origin), is optional and relevant only for SYSTEM=RECTANGULAR and SYSTEM=Z RECTANGULAR. The default location of the origin, *c*, is the first node of the element (local node number 1).

3. Local node number of point *c*.

Second line:

1. Local direction about which the additional rotation or rotations are given. The default is the local 1-direction. For shell, membrane, and cohesive elements this direction should have a nonzero component in the direction of the normal to the surface.
2. Additional rotation α defined by either a single scalar value or by a distribution. A local coordinate system defined with a distribution can only be used for solid continuum elements and shell elements. The additional rotation (in degrees) is applied to both directions orthogonal to the specified local direction. The default is zero degrees.

Third line when the LOCAL DIRECTIONS parameter is included:

1. *X*-component of the first local material direction with respect to the orthonormal system at the material point.

2. *Y*-component of the first local material direction with respect to the orthonormal system at the material point.
3. *Z*-component of the first local material direction with respect to the orthonormal system at the material point.

Repeat the above data line to define additional local directions, as needed, with each direction on a separate line.

To define an orientation using a user subroutine (SYSTEM=USER):

No data lines are used with this option when SYSTEM=USER is specified. Instead, user subroutine **ORIENT** must be used to define the orientation.

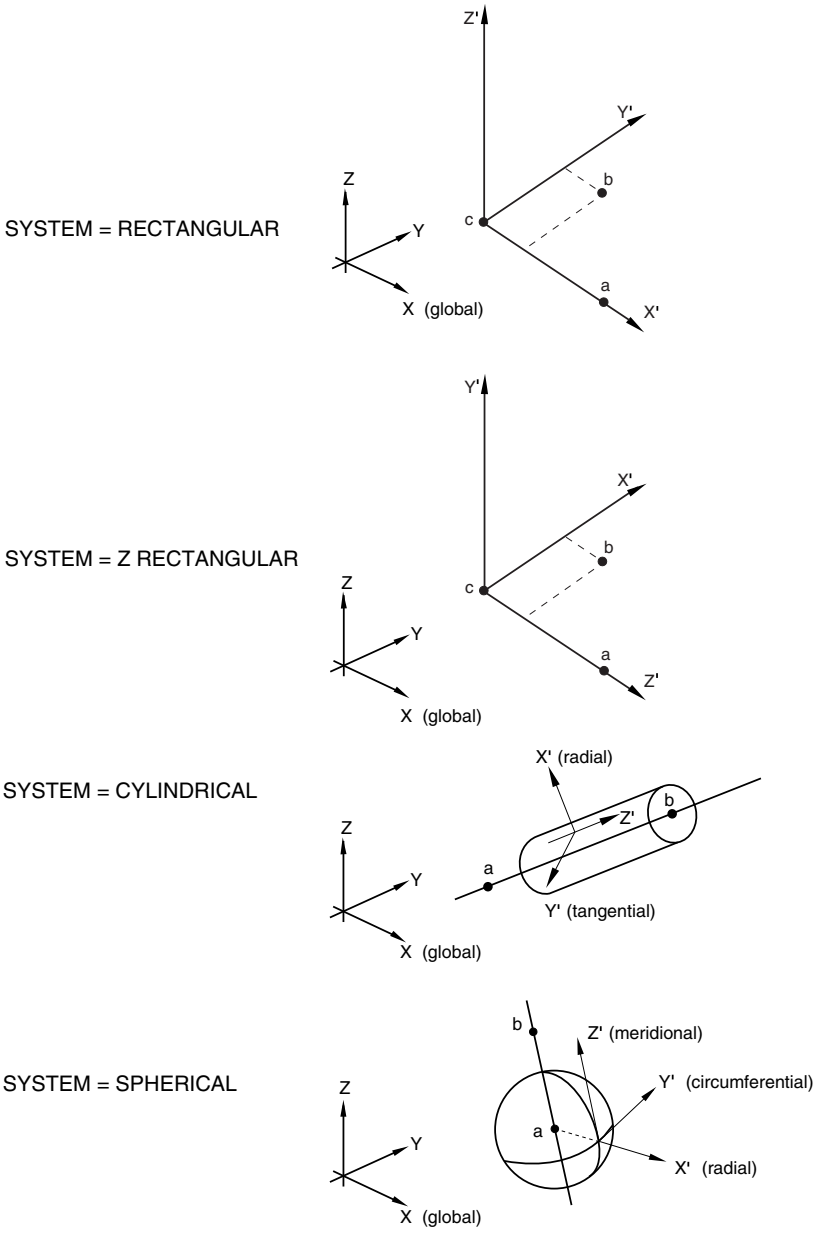


Figure 15.1–1 Orientation systems.

15.2 *ORNL: Specify constitutive model developed by Oak Ridge National Laboratory.

This option is used to provide plasticity and creep calculations for type 304 and 316 stainless steel according to the specification in Nuclear Standard NEF 9–5 T, "Guidelines and Procedures for Design of Class I Elevated Temperature Nuclear System Components." It can be used only with the *PLASTIC option and/or the *CREEP, LAW=STRAIN option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "ORNL – Oak Ridge National Laboratory constitutive model," Section 20.2.12 of the Abaqus Analysis User's Manual

Optional parameters:

A

Set this parameter equal to the saturation rates for kinematic shift caused by creep strain, as defined by Equation (15) of Section 4.3.3–3 of the Nuclear Standard. The default value is 0.3, as per that section of the Standard. Set A=0.0 to use the 1986 revision of the Standard.

H

Set this parameter equal to the rate of kinematic shift with respect to creep strain [Equation (7) of Section 4.3.2–1 of the Nuclear Standard]. Set H=0.0 to use the 1986 revision of the Standard. If this parameter is omitted, the value of H is determined according to Section 4.3.3–3 of the 1981 revision of the Standard.

MATERIAL

Set MATERIAL=SS to use the hardening law appropriate to either type 304 or type 316 stainless steel. This is the only option presently available and, thus, the default.

RESET

Include this parameter to invoke the optional α reset procedure described in Section 4.3.5 of the Nuclear Standard. If this parameter is omitted, the α reset procedure is not used.

There are no data lines associated with this option.

15.3 *OUTPUT: Define output requests to the output database.

This option is used to write contact, element, energy, nodal, or diagnostic output to the output database. In an Abaqus/Standard analysis it is also used to write modal or radiation output to the output database. In an Abaqus/Explicit analysis it is also used to write incrementation output to the output database. The *CONTACT OUTPUT, *ELEMENT OUTPUT, *ENERGY OUTPUT, *INCREMENTATION OUTPUT, *MODAL OUTPUT, *NODE OUTPUT, and/or *RADIATION OUTPUT options can be used in conjunction with this option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

Using *OUTPUT in an Abaqus/Standard analysis

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual
- "Overview of job diagnostics," Section 39.1 of the Abaqus/CAE User's Manual
- *CONTACT OUTPUT
- *ELEMENT OUTPUT
- *ENERGY OUTPUT
- *MODAL OUTPUT
- *NODE OUTPUT
- *RADIATION OUTPUT
- *TIME POINTS

One of the following mutually exclusive parameters is required:

DIAGNOSTICS

Set DIAGNOSTICS=YES (default) to indicate that detailed diagnostic information should be written to the output database. Set DIAGNOSTICS=NO to suppress the output.

FIELD

Include this parameter to indicate that the output requests used in conjunction with the *OUTPUT option will be written to the output database as field-type output.

HISTORY

Include this parameter to indicate that the output requests used in conjunction with the *OUTPUT option will be written to the output database as history-type output.

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written to the output database at the last increment of each step. Set FREQUENCY=0 to suppress the output.

If this parameter and the NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are omitted, output will be written at every increment of the analysis for all procedure types except *DYNAMIC and *MODAL DYNAMIC; output will be written every 10 increments for these procedure types.

The FREQUENCY, NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive.

MODE LIST

Include this parameter to indicate that a list of eigenmodes for which output is desired will be listed on the data lines. This parameter is valid only in a *FREQUENCY, *COMPLEX FREQUENCY, or *BUCKLE procedure, and if the FIELD parameter is included.

NAME

Set this parameter equal to the name associated with this output definition.

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the output database states are to be written.

If this parameter and the FREQUENCY, TIME INTERVAL, and TIME POINTS parameters are omitted, output will be written at every increment of the analysis for all procedure types except *DYNAMIC and *MODAL DYNAMIC; output will be written every 10 increments for these procedure types.

The FREQUENCY, NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive.

TIME MARKS

Set TIME MARKS=YES (default) to write results at the exact times dictated by the NUMBER INTERVAL, TIME INTERVAL, or TIME POINTS parameter.

Set TIME MARKS=NO to write results to the output database at the increment ending immediately after the time dictated by the NUMBER INTERVAL, TIME INTERVAL, or TIME POINTS parameter.

TIME POINTS

Set this parameter equal to the name of the *TIME POINTS option that defines the time points at which output is to be written.

If this parameter and the FREQUENCY, NUMBER INTERVAL, and TIME INTERVAL parameters are omitted, output will be written at every increment of the analysis for all procedure types except *DYNAMIC and *MODAL DYNAMIC; output will be written every 10 increments for these procedure types.

The FREQUENCY, NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive.

The following parameters are optional and valid only if the FIELD or HISTORY parameter is included:

OP

Set OP=NEW (default) to indicate that all output database requests defined in previous steps should be removed. New output database requests can be defined.

Set OP=ADD to indicate that the output request being defined should be added to the output requests defined in previous steps.

Set OP=REPLACE to indicate that this output request should replace an output request of the same type (e.g., FIELD) and frequency defined in a previous step. If there is no matching request, this output request will be interpreted as OP=ADD.

TIME INTERVAL

Set this parameter equal to the time interval at which the output states are to be written.

If this parameter and the FREQUENCY, NUMBER INTERVAL, and TIME POINTS parameters are omitted, output will be written at every increment of the analysis for all procedure types except *DYNAMIC and *MODAL DYNAMIC; output will be written every 10 increments for these procedure types.

The FREQUENCY, NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive.

VARIABLE

Set VARIABLE=ALL to indicate that all variables applicable to this procedure and material type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default output variables for the current procedure type should be written to the output database. Additional output requests can be defined with the output options used in conjunction with the *OUTPUT option, listed previously.

If this parameter is omitted, only the variables requested for output with the individual output options will be written to the output database.

The output behavior of the variables included in ALL or PRESELECT cannot be controlled by the parameters or data lines of the individual output options. To get the desired output behavior of a particular variable in an individual output option, the particular variable must be listed in that individual output option.

The following parameter is optional and valid only if the HISTORY parameter is included:

SENSOR

Include this parameter to associate this history output request with a sensor definition. The name of the associated sensor is given by the NAME parameter.

Data lines to list desired eigenmodes if the MODE LIST parameter is included:

First line:

1. Specify a list of desired eigenmodes.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Using *OUTPUT in an Abaqus/Explicit analysis

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- "Abaqus/Explicit output variable identifiers," Section 4.2.2 of the Abaqus Analysis User's Manual
- "Overview of job diagnostics," Section 39.1 of the Abaqus/CAE User's Manual
- *CONTACT OUTPUT
- *ELEMENT OUTPUT
- *ENERGY OUTPUT
- *INCREMENTATION OUTPUT
- *NODE OUTPUT
- *TIME POINTS

One of the following mutually exclusive parameters is required:

DIAGNOSTICS

Set DIAGNOSTICS=YES (default) to indicate that detailed diagnostic information should be written to the output database. Set DIAGNOSTICS=NO to suppress the output.

FIELD

Include this parameter to indicate that the output requests used in conjunction with the *OUTPUT option will be written to the output database as field-type output.

HISTORY

Include this parameter to indicate that the output requests used in conjunction with the *OUTPUT option will be written to the output database as history-type output.

Optional parameter:

NAME

Set this parameter equal to the name associated with this output definition.

The following parameters are optional and valid only if the FIELD parameter is included:

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the output database states are to be written. Abaqus/Explicit will always write the results at the beginning of the step. For example, if NUMBER INTERVAL=10, Abaqus/Explicit will write 11 output database states consisting of the values at the beginning of the step and the values at the end of 10 intervals throughout the step. The value of this parameter must be a positive integer or zero. A value of zero suppresses all output. If this parameter is omitted, its value will be set to 20.

The NUMBER INTERVAL and TIME POINTS parameters are mutually exclusive.

TIME MARKS

Set TIME MARKS=NO (default) to write results to the output database at the increment ending immediately after the time dictated by the NUMBER INTERVAL or TIME POINTS parameter.

Set TIME MARKS=YES to write results at the exact times dictated by the NUMBER INTERVAL or TIME POINTS parameter.

TIME MARKS=YES cannot be used when either the FIXED TIME INCREMENTATION or DIRECT USER CONTROL parameter is included on the *DYNAMIC option.

TIME POINTS

Set this parameter equal to the name of the *TIME POINTS option that defines the time points at which output is to be written. If this parameter and the NUMBER INTERVAL parameter are omitted, field output will be written at 20 equally spaced intervals throughout the step.

The NUMBER INTERVAL and TIME POINTS parameters are mutually exclusive.

The following parameters are optional and valid only if the HISTORY parameter is included:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written to the output database at the last increment of each step. Set FREQUENCY=0 to suppress the output. If both this parameter and the TIME INTERVAL parameter are omitted, history output will be written at 200 equally spaced intervals throughout the step.

The FREQUENCY and TIME INTERVAL parameters are mutually exclusive.

SENSOR

Include this parameter to associate this history output request with a sensor definition. The name of the associated sensor is given by the NAME parameter.

The following parameters are optional and valid only if the FIELD or HISTORY parameter is included:

FILTER

Set this parameter equal to the name of the *FILTER option to be used to filter the element and nodal field output or the element, nodal, contact, fastener interaction, or integrated history output.

Set FILTER=ANTIALIASING to filter the data based on the time interval that is specified; in this case the filter does not need to be defined in the model data. The antialiasing filter cannot be used with the FREQUENCY parameter for history output.

OP

Set OP=NEW (default) to indicate that all output database requests defined in previous steps should be removed. New output database requests can be defined.

Set OP=ADD to indicate that the output request being defined should be added to the output requests defined in previous steps.

Set OP=REPLACE to indicate that this output request should replace an output request of the same type (e.g., FIELD) and frequency defined in a previous step. If there is no matching request, this output request will be interpreted as OP=ADD.

TIME INTERVAL

Set this parameter equal to the time interval at which the output states are to be written.

For field output Abaqus/Explicit will always write the output at the beginning of the step. If both this parameter and the NUMBER INTERVAL parameter are omitted, field output will be written at 20 equally spaced intervals throughout the step. The NUMBER INTERVAL, TIME INTERVAL, and TIME POINTS parameters are mutually exclusive for field data.

For history output Abaqus/Explicit will always write the data values at the beginning and end of the step. If both this parameter and the FREQUENCY parameter are omitted, history output will be written at 200 equally spaced intervals throughout the step. The FREQUENCY and TIME INTERVAL parameters are mutually exclusive for history data.

VARIABLE

Set VARIABLE=ALL to indicate that all variables applicable to this procedure and material type should be written to the output database.

Set VARIABLE=PRESELECT to indicate that the default output variables for the current procedure type should be written to the output database. Additional output requests can be defined with the output options used in conjunction with the *OUTPUT option, listed previously.

If this parameter is omitted, only the variables requested for output with the individual output options will be written to the output database.

The output behavior of the variables included in ALL or PRESELECT cannot be controlled by the parameters or data lines of the individual output options. To get the desired output behavior of a particular variable in an individual output option, the particular variable must be listed in that individual output option.

There are no data lines associated with this option in Abaqus/Explicit.

16. P, Q

16.1 *PARAMETER: Define parameters for input parametrization.

This option is used to define parameters that can be used in place of Abaqus input quantities.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly, Model, Step

References:

- "Parametric input," Section 1.4.1 of the Abaqus Analysis User's Manual
- "Parametric shape variation," Section 2.1.2 of the Abaqus Analysis User's Manual

Optional keyword parameters to define tabularly dependent parameters (if any are specified, they must all be specified):

DEPENDENT

Set this keyword parameter equal to the list of dependent parameters being defined in this option. The list must be given inside parentheses as parameter names separated by commas; for example, `(depPar1, depPar2, depPar3)`.

INDEPENDENT

Set this keyword parameter equal to the list of independent parameters used in this option. The list must be given inside parentheses as parameter names separated by commas; for example, `(indPar1, indPar2, indPar3)`.

TABLE

Set this keyword parameter equal to the name of the parameter dependence table, defined by the *PARAMETER DEPENDENCE option, that defines the relationship between the dependent and independent parameters in this option.

Data lines to define independent or expressionally dependent parameters if the DEPENDENT, INDEPENDENT, and TABLE keyword parameters are omitted:

First line:

1. Python expression assigning a value to a parameter.

Repeat this data line as often as necessary to define independent and expressionally dependent parameters. The data given on this data line cannot be parameterized.

16.2 *PARAMETER DEPENDENCE: Define dependence table for tabularly dependent parameters.

This option is used to define the dependence table that specifies the relationship between tabularly dependent and independent parameters.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance, Assembly, Model, Step

References:

- "Parametric input," Section 1.4.1 of the Abaqus Analysis User's Manual
- "Parametric shape variation," Section 2.1.2 of the Abaqus Analysis User's Manual

Required parameters:

NUMBER VALUES

Set this keyword parameter equal to the number of values in each line of the parameter dependence table. This number must be equal to the sum of the number of dependent and the number of independent parameters for which this table is used.

TABLE

Set this keyword parameter equal to the name of the table being defined in this option.

Data lines to define the parameter dependence table:

First line:

1. Values in the first row of the dependence table. Give the dependent and independent parameter data values that are to be interpolated. The dependent parameter values must precede the independent parameter values. The total number of entries in the row must equal the value of the NUMBER VALUES parameter.

Repeat this data line as often as necessary to define additional rows of the parameter dependence table. The data given on this data line cannot be parameterized.

16.3 *PARAMETER SHAPE VARIATION: Define parametric shape variations.

This option is used to define parametric shape variations.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Parametric shape variation," Section 2.1.2 of the Abaqus Analysis User's Manual
- "Design sensitivity analysis," Section 16.1.1 of the Abaqus Analysis User's Manual

Required parameter:

PARAMETER

Set this keyword parameter equal to the name of the parameter to which the shape variation data refer. If this parameter is also a design parameter, the shape variations are used to define the design gradients of the nodal coordinates for design sensitivity analysis.

Optional parameters (mutually exclusive—if neither parameter is specified, Abaqus assumes that the shape variation data will be entered directly on the data lines):

FILE

Set this parameter equal to the name of the results file from a previous Abaqus/Standard analysis containing either the mode shapes from a *BUCKLE or *FREQUENCY analysis or the nodal displacements from a *STATIC analysis. This option cannot be used for models defined in terms of an assembly of part instances.

INPUT

Set this parameter equal to the name of the alternate input file containing the shape variation data. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names.

Required parameter if the FILE parameter is used:

STEP

Set this parameter equal to the step number (in the analysis whose results file is being used as input to this option) from which the modal or displacement data are to be read.

Optional parameters if the FILE parameter is used:

INC

Set this parameter equal to the increment number (in the analysis whose results file is being used as input to this option) from which the displacement data are to be read. If this parameter is omitted, Abaqus will read the data from the last increment available for the specified step on the results file.

MODE

Set this parameter equal to the mode number (in the analysis whose results file is being used as input to this option) from which the modal data are to be read. If this parameter is omitted, Abaqus will read the data from the first mode available for the specified step on the results file.

NSET

Set this parameter equal to the node set to which the shape variation values are to be applied. If this parameter is omitted, the shape variation will be applied to all nodes in the model.

Optional parameter if the FILE parameter is omitted:

SYSTEM

Set SYSTEM=R (default) to specify the shape variation as values of Cartesian coordinates. Set SYSTEM=C to specify the shape variation as values of cylindrical coordinates. Set SYSTEM=S to specify the shape variation as values of spherical coordinates. See Figure 16.3–1.

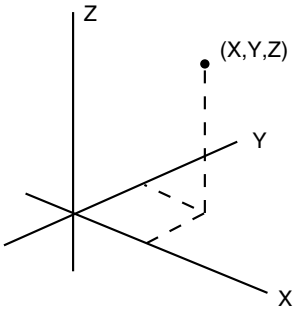
The SYSTEM parameter is entirely local to this option and should not be confused with the *SYSTEM option. As the data lines are read, the shape variation values specified are transformed to the global rectangular Cartesian coordinate system. This transformation requires that the object be centered about the origin of the global coordinate system; i.e., the *SYSTEM option should be off when specifying shape variations as values using either cylindrical or spherical coordinates. The details of how the shape variation is computed in particular coordinate systems are given in "Parametric shape variation," Section 2.1.2 of the Abaqus Analysis User's Manual.

Data lines to define the shape variation if the FILE and INPUT parameters are omitted:

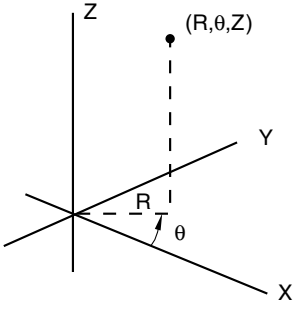
First line:

1. Node number or node set.
2. Component of shape variation in the first coordinate direction.
3. Component of shape variation in the second coordinate direction.
4. Component of shape variation in the third coordinate direction.
5. Shape variation in the first normal component.
6. Shape variation in the second normal component.
7. Shape variation in the third normal component.

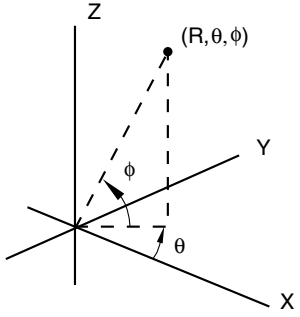
Repeat this data line as often as necessary to define the shape variation. The data given on this data line cannot be parameterized.



Rectangular Cartesian
(SYSTEM=R)
(default)



Cylindrical
(SYSTEM=C)



Spherical
(SYSTEM=S)

(θ and ϕ are given in degrees)

Figure 16.3–1 Coordinate systems.

16.4 *PART: Begin a part definition.

This option is used to begin a part definition. It must be used in conjunction with the *ASSEMBLY, *END PART, and *INSTANCE options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Part module

References:

- "Defining an assembly," Section 2.9.1 of the Abaqus Analysis User's Manual
- *END PART

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the part.

There are no data lines associated with this option.

16.5 *PERIODIC: Define periodic symmetry for a cavity radiation heat transfer analysis.

This option is used to define cavity symmetry by periodic repetition in a given direction. It can be used only following the *RADIATION SYMMETRY option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual
- *RADIATION SYMMETRY

Required parameter:

TYPE

Set TYPE=2D to create a cavity composed of the cavity surface defined in the model and a series of similar images generated by its repetition according to a two-dimensional distance vector. The repeated images are bounded by lines parallel to line ab (see Figure 16.5–1). The distance vector must be defined so that it points away from line ab and into the domain of the model. This option can be used only for two-dimensional cases.

Set TYPE=3D to create a cavity composed of the cavity surface defined in the model and a series of similar images generated by its repetition according to a three-dimensional distance vector. The repeated images are bounded by planes parallel to plane abc (see Figure 16.5–2). The distance vector must be defined so that it points away from plane abc and into the domain of the model. This option can be used only for three-dimensional cases.

Set TYPE=ZDIR to create a cavity composed of the cavity surface defined in the model and a series of similar images generated by its repetition in the z-direction. The repeated images are bounded by lines of constant z-coordinate (see Figure 16.5–3). The z-distance vector must be defined so that it points away from the z-constant periodic symmetry reference line and into the domain of the model. This option can be used only for axisymmetric cases.

Optional parameter:

NR

Set this parameter equal to the number of repetitions used in the numerical calculation of the cavity viewfactors resulting from the periodic symmetry. The result of this symmetry is a cavity composed of the cavity surface defined in the model plus twice NR similar images, since the periodic symmetry is assumed to apply both in the positive and negative directions of the distance vector. The default value is NR=2.

Data line to define periodic symmetry of a two-dimensional cavity (TYPE=2D):

First (and only) line:

1. x-coordinate of point a (see Figure 16.5–1).
2. y-coordinate of point a.
3. x-coordinate of point b.
4. y-coordinate of point b.
5. x-component of periodic distance vector.
6. y-component of periodic distance vector.

Data lines to define periodic symmetry of a three-dimensional cavity (TYPE=3D):

First line:

1. x-coordinate of point a (see Figure 16.5–2).
2. y-coordinate of point a.
3. z-coordinate of point a.
4. x-coordinate of point b.
5. y-coordinate of point b.
6. z-coordinate of point b.

Second line:

1. x-coordinate of point c.
2. y-coordinate of point c.
3. z-coordinate of point c.
4. x-component of periodic distance vector.
5. y-component of periodic distance vector.
6. z-component of periodic distance vector.

Data line to define periodic symmetry of an axisymmetric cavity (TYPE=ZDIR):

First (and only) line:

1. z-coordinate of periodic symmetry reference line (see Figure 16.5–3).
2. z-component of periodic distance vector.

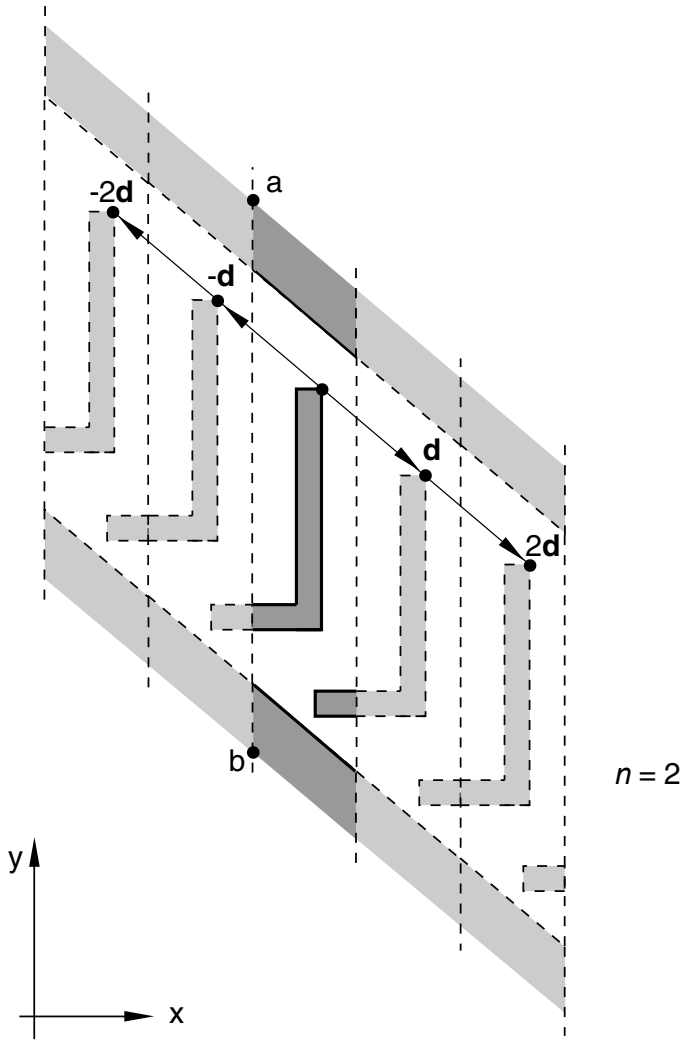


Figure 16.5–1 *PERIODIC, TYPE=2D option.

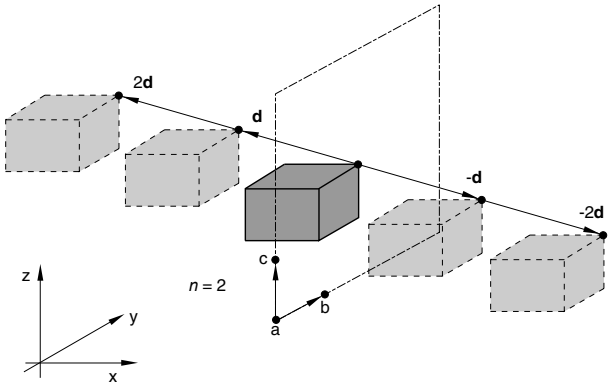


Figure 16.5–2 *PERIODIC, TYPE=3D option.

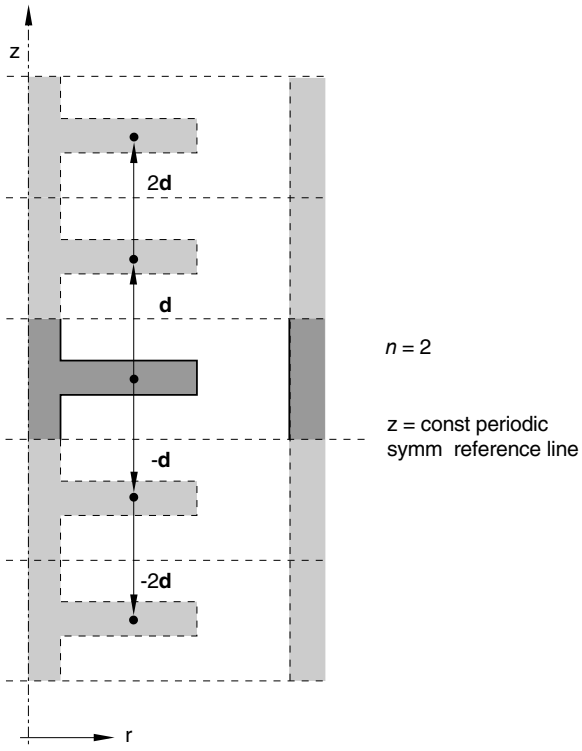


Figure 16.5–3 *PERIODIC, TYPE=ZDIR option.

16.6 *PERMEABILITY: Define permeability for pore fluid flow.

This option is used to define permeability for pore fluid flow in problems involving seepage.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Permeability," Section 23.7.2 of the Abaqus Analysis User's Manual

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the permeability. If this parameter is omitted, it is assumed that the permeability is independent of field variables. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

This parameter can be used only in conjunction with TYPE=ISOTROPIC, ORTHOTROPIC, or ANISOTROPIC.

TYPE

Set TYPE=ISOTROPIC (default) to define fully saturated isotropic permeability. Set TYPE=ORTHOTROPIC to define fully saturated orthotropic permeability. Set TYPE=ANISOTROPIC to define fully saturated anisotropic permeability.

Set TYPE=SATURATION to define $k_s(s)$; this must be a repeated use of the option for the same material and must follow the definition of fully saturated permeability. The definition must give k_s for $0.0 < s \leq 1.0$, with $k_s = 1.0$ at $s = 1.0$.

Set TYPE=VELOCITY to define $\beta(e)$; this must be a repeated use of the option for the same material and must follow the definition of fully saturated permeability.

Required parameter when fully saturated material properties are defined:

SPECIFIC

Set this parameter equal to the specific weight of the wetting liquid, γ_w (units of FL^{-3}). The actual specific weight value must be given, and the GRAV distributed load type must be used to apply the gravitational loading if a total pressure solution is required (see "Coupled pore fluid diffusion and

stress analysis," Section 6.8.1 of the Abaqus Analysis User's Manual, for a discussion of total and excess pressure solutions).

Data lines to define fully saturated isotropic permeability (TYPE=ISOTROPIC):

First line:

1. k. (Units of LT^{-1}.)
2. Void ratio, e.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define fully saturated orthotropic permeability (TYPE=ORTHOTROPIC):

First line:

1. k_{11}. (Units of LT^{-1}.)
2. k_{22}.
3. k_{33}.
4. Void ratio, e.
5. Temperature, θ.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define fully saturated anisotropic permeability (TYPE=ANISOTROPIC):

First line:

1. k_{11}. (Units of LT^{-1}.)
2. k_{12}.
3. k_{22}.

4. k_{13}.
5. k_{23}.
6. k_{33}.
7. Void ratio, *e*.
8. Temperature, θ.

Subsequent lines (only needed if the DEPENDENCIES parameter is specified):

1. First field variable.
2. Second field variable.
3. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the variation.

Data lines to define the dependence of permeability on saturation of the wetting liquid, $k_s(s)$ (TYPE=SATURATION):

First line:

1. k_s. (Dimensionless.)
2. Saturation, *s*. (Dimensionless.)

Repeat this data line as often as necessary to define the variation. The table must provide $k_s = 1.0$ at $s = 1.0$.

Data lines to define the velocity coefficient (TYPE=VELOCITY):

First line:

1. β. (Units of TL^{-1}.) Only $\beta > 0.0$ is allowed.
2. Void ratio, *e*.

Repeat this data line as often as necessary to define the variation.

16.7 *PHYSICAL CONSTANTS: Specify physical constants.

This option is used to define physical constants necessary for an analysis; since Abaqus has no built-in units, no default values are provided. If a physical constant required for the analysis is not given, Abaqus will issue a fatal error message. The units used for the constants must be consistent with the remaining input data.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Model attribute

References:

- "Uncoupled heat transfer analysis," Section 6.5.2 of the Abaqus Analysis User's Manual
- "Mass diffusion analysis," Section 6.9.1 of the Abaqus Analysis User's Manual
- "Modeling fluid-filled cavities," Section 11.5.1 of the Abaqus Analysis User's Manual
- "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual
- "Hydrostatic fluid models," Section 23.4.1 of the Abaqus Analysis User's Manual
- "Diffusivity," Section 23.5.1 of the Abaqus Analysis User's Manual
- "Solubility," Section 23.5.2 of the Abaqus Analysis User's Manual
- "Thermal contact properties," Section 33.2.1 of the Abaqus Analysis User's Manual
- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

ABSOLUTE ZERO

Set this parameter equal to the absolute zero on the temperature scale chosen. For example, if the analysis uses temperature in degrees Celsius, set ABSOLUTE ZERO=−273.15.

STEFAN BOLTZMANN

Set this parameter equal to the Stefan Boltzmann constant. For example, STEFAN BOLTZMANN=5.669×10^{-8} joule per sec m^2 $kelvin^4$ in SI units.

UNIVERSAL GAS CONSTANT

Set this parameter equal to the universal gas constant. For example, UNIVERSAL GAS CONSTANT=8.31434 joule per mole kelvin in SI units.

SPL REFERENCE PRESSURE

Set this parameter equal to the reference pressure used for computing sound pressure level. For example, SPL REFERENCE PRESSURE=20. micro pascals for air in SI units.

There are no data lines associated with this option.

16.8 *PIEZOELECTRIC: Specify piezoelectric material properties.

This option is used to define the piezoelectric properties of a material.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Piezoelectric behavior," Section 23.6.2 of the Abaqus Analysis User's Manual

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of the piezoelectric properties. If this parameter is omitted, the piezoelectric properties are assumed not to depend on any field variables but may still depend on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

TYPE

Set TYPE=S (default) to specify stress material coefficients for the piezoelectric property. Set TYPE=E to specify strain material coefficients for the piezoelectric property.

Data lines to define the piezoelectric stress coefficient matrix (TYPE=S):

First line:

1. $e^{\varphi}_{1\,11}$. (Units of $\mathrm{FL}^{-1}\varphi^{-1}$.)
2. $e^{\varphi}_{1\,22}$.
3. $e^{\varphi}_{1\,33}$.
4. $e^{\varphi}_{1\,12}$.
5. $e^{\varphi}_{1\,13}$.
6. $e^{\varphi}_{1\,23}$.
7. $e^{\varphi}_{2\,11}$.
8. $e^{\varphi}_{2\,22}$.

Second line:

1. $e^{\varphi}_{2\,33}$.
2. $e^{\varphi}_{2\,12}$.
3. $e^{\varphi}_{2\,13}$.
4. $e^{\varphi}_{2\,23}$.
5. $e^{\varphi}_{3\,11}$.
6. $e^{\varphi}_{3\,22}$.
7. $e^{\varphi}_{3\,33}$.
8. $e^{\varphi}_{3\,12}$.

Third line:

1. $e^{\varphi}_{3\,13}$.
2. $e^{\varphi}_{3\,23}$.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the piezoelectric properties as a function of temperature and other predefined field variables.

Data lines to define the piezoelectric strain coefficient matrix (TYPE=E; for the shear components these coefficients relate the engineering, and not the tensorial, shear strain components to the components of the potential gradient vector):

First line:

1. $d^{\varphi}_{1\,11}$. (Units of $\mathrm{L}\varphi^{-1}$.)
2. $d^{\varphi}_{1\,22}$.
3. $d^{\varphi}_{1\,33}$.
4. $d^{\varphi}_{1\,12}$.
5. $d^{\varphi}_{1\,13}$.
6. $d^{\varphi}_{1\,23}$.
7. $d^{\varphi}_{2\,11}$.
8. $d^{\varphi}_{2\,22}$.

Second line:

1. $d^{\varphi}_{2\,33}$.
2. $d^{\varphi}_{2\,12}$.
3. $d^{\varphi}_{2\,13}$.
4. $d^{\varphi}_{2\,23}$.
5. $d^{\varphi}_{3\,11}$.
6. $d^{\varphi}_{3\,22}$.
7. $d^{\varphi}_{3\,33}$.
8. $d^{\varphi}_{3\,12}$.

Third line:

1. $d^{\varphi}_{3\,13}$.
2. $d^{\varphi}_{3\,23}$.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the piezoelectric properties as a function of temperature and other predefined field variables.

16.9 *PIPE-SOIL INTERACTION: Specify element properties for pipe-soil interaction elements.

This option is used to define properties for pipe-soil interaction elements. The *PIPE-SOIL STIFFNESS option must follow immediately after this option.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Pipe-soil interaction elements," Section 29.13.1 of the Abaqus Analysis User's Manual
- "Pipe-soil interaction element library," Section 29.13.2 of the Abaqus Analysis User's Manual
- *PIPE-SOIL STIFFNESS

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the pipe-soil interaction elements for which properties are being defined.

Optional parameter:

ORIENTATION

Set this parameter equal to the name of the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) that gives the orientation of the local system used for material calculations.

There are no data lines associated with this option.

16.10 *PIPE-SOIL STIFFNESS: Define constitutive behavior for pipe-soil interaction elements.

This option is used to define the constitutive behavior for pipe-soil interaction elements. It can be used only in conjunction with the *PIPE-SOIL INTERACTION option. Repeat the option as needed to define behavior in the different local directions.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Pipe-soil interaction elements," Section 29.13.1 of the Abaqus Analysis User's Manual
- "Pipe-soil interaction element library," Section 29.13.2 of the Abaqus Analysis User's Manual
- "UMAT," Section 1.1.36 of the Abaqus User Subroutines Reference Manual
- *PIPE-SOIL INTERACTION

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material property values. If this parameter is omitted, it is assumed that the properties are independent of field variables. See "Using the DEPENDENCIES parameters to define field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual for more information.

DIRECTION

Set this parameter equal to the direction in the local orientation system for which the behavior is defined. The DIRECTION parameter can be set equal to a label or to a numerical value. Omit the DIRECTION parameter to define an isotropic model. The DIRECTION parameter must be used to define a constitutive model using the ASCE formulae.

Set DIRECTION=AXIAL (or DIRECTION=1) to specify behavior along the first local direction.

Set DIRECTION=VERTICAL (or DIRECTION=2) to specify behavior along the second local direction.

Set DIRECTION=HORIZONTAL (or DIRECTION=3) to specify behavior along the third local direction.

TYPE

Set TYPE=LINEAR (default) to define a linear constitutive model.

Set TYPE=NONLINEAR to define a nonlinear constitutive model.

Set TYPE=CLAY to define a constitutive model using the ASCE formulae for clay. This parameter must be used in conjunction with the DIRECTION parameter.

Set TYPE=SAND to define a constitutive model using the ASCE formulae for sand. This parameter must be used in conjunction with the DIRECTION parameter.

Set TYPE=USER to indicate the constitutive behavior is defined in user subroutine **UMAT**.

The following optional parameters can be used only in combination with TYPE=USER:

PROPERTIES

Set this parameter equal to the number of property values needed as data in user subroutine **UMAT**. The default is PROPERTIES=0.

VARIABLES

Set this parameter equal to the number of solution-dependent variables that must be stored for the material calculations in user subroutine **UMAT**. The default is VARIABLES=1.

Data lines to define linear constitutive behavior (TYPE=LINEAR):

First line:

1. Interaction stiffness (force per relative displacement) corresponding to positive relative displacements, K_p.
2. Interaction stiffness (force per relative displacement) corresponding to negative relative displacements, K_n. The default is $K_n = K_p$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the stiffness as a function of temperature and other predefined field variables.

Data lines to define nonlinear constitutive behavior (TYPE=NONLINEAR):

First line:

1. Force per unit length along pipeline.
2. Relative displacement.

3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the force per unit length as a function of relative displacement, temperature, and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for sand in the axial direction (TYPE=SAND, DIRECTION=AXIAL):

First line:

1. Effective unit weight of soil, $\bar{\gamma}$.
2. Coefficient of soil pressure at rest, K_0.
3. Interface angle of friction, δ. Give the value in degrees.
4. External pipe diameter, D.
5. Ultimate relative displacement, $\bar{\varepsilon}_a$.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for clay in the axial direction (TYPE=CLAY, DIRECTION=AXIAL):

First line:

1. Undrained soil shear strength, S.
2. Empirical adhesion factor, α.
3. External pipe diameter, D.
4. Ultimate relative displacement, $\bar{\varepsilon}_a$.
5. Temperature.
6. First field variable.

7. Second field variable.
8. Third field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for sand in the vertical direction (TYPE=SAND, DIRECTION=VERTICAL):

First line:

1. Total unit weight of soil, γ.
2. Bearing capacity factor, N_q.
3. Bearing capacity factor, N_γ.
4. Vertical uplift factor, N_{qv}.
5. Ultimate relative displacement for downward motion, $\bar{\varepsilon}_{vp}$.
6. Ultimate relative displacement for upward motion, $\bar{\varepsilon}_{vn}$.
7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for clay in the vertical direction (TYPE=CLAY, DIRECTION=VERTICAL):

First line:

1. Bearing capacity factor, N_c.
2. Vertical uplift factor, N_{cv}.
3. Ultimate relative displacement for downward motion, $\bar{\varepsilon}_{vp}$.
4. Ultimate relative displacement for upward motion, $\bar{\varepsilon}_{vn}$.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for sand in the horizontal direction (TYPE=SAND, DIRECTION=HORIZONTAL):

First line:

1. Horizontal bearing capacity factor, N_{qh}.
2. Ultimate relative displacement, $\bar{\varepsilon}_h$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines to define constitutive behavior using the ASCE formula for clay in the horizontal direction (TYPE=CLAY, DIRECTION=HORIZONTAL):

First line:

1. Horizontal bearing capacity factor, N_{ch}.
2. Ultimate relative displacement, $\bar{\varepsilon}_h$.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables per line.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define parameters for the ASCE formulae as a function of temperature and other predefined field variables.

Data lines if the constitutive behavior is defined in user subroutine `UMAT` (TYPE=USER):

First line:

1. Enter the data to be used as properties in user subroutine **`UMAT`**.

Repeat this data line as often as necessary to define properties required in **`UMAT`**. Enter eight values per line.

16.11 *PLANAR TEST DATA: Used to provide planar test (or pure shear) data (compression and/or tension).

This option is used to provide planar test (or pure shear) data. It can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, and the *MULLINS EFFECT option. This type of test does not define the hyperelastic material constants fully; at the least, uniaxial or biaxial test data should also be given.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Using planar test data to define a hyperelastic material

References:

- "Hyperelastic behavior of rubberlike materials," Section 19.5.1 of the Abaqus Analysis User's Manual
- *HYPERELASTIC

Optional parameter:

SMOOTH

Include this parameter to apply a smoothing filter to the stress-strain data. If the parameter is omitted, no smoothing is performed.

Set this parameter equal to the number n such that $2n + 1$ is equal to the total number of data points in the moving window through which a cubic polynomial is fit using the least-squares method. n should be larger than 1. The default is SMOOTH=3.

Optional parameter when the *PLANAR TEST DATA option is used in conjunction with the *HYPERELASTIC, MARLOW option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to specify planar test data for hyperelasticity other than the Marlow model (the nominal strains must be arranged in either ascending or descending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_S.
2. Nominal strain in the direction of loading, ϵ_S.

Repeat this data line as often as necessary to give the stress-strain data.

Data lines to specify planar test data for the Marlow model (the nominal strains must be arranged in ascending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_S.
2. Nominal strain, ϵ_S.
3. Nominal lateral strain, ϵ_3. Not needed if the POISSON parameter is specified on the *HYPERELASTIC option or if the *VOLUMETRIC TEST DATA option is used.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. Nominal strains and nominal stresses must be given in ascending order.

Using planar test data to define an elastomeric foam

References:

- "Hyperelastic behavior in elastomeric foams," Section 19.5.2 of the Abaqus Analysis User's Manual
- *HYPERFOAM

There are no parameters associated with this option.

Data lines to specify planar test data for a hyperfoam:

First line:

1. Nominal stress, T_L.
2. Nominal strain in the direction of loading, ϵ_P.
3. Nominal transverse strain, ϵ_3. Default is zero. Not needed if the POISSON parameter is specified on the *HYPERFOAM option.

Repeat this data line as often as necessary to give the stress-strain data.

Using planar test data to define the Mullins effect material model

References:

- "Mullins effect in rubberlike materials," Section 19.6.1 of the Abaqus Analysis User's Manual
- "Energy dissipation in elastomeric foams," Section 19.6.2 of the Abaqus Analysis User's Manual
- *MULLINS EFFECT

There are no parameters associated with this option.

Data lines to specify planar test data for defining the unloading-reloading response of the Mullins effect material model:

First line:

1. Nominal stress, T_L.
2. Nominal strain in the direction of loading, ϵ_P.

Repeat this data line as often as necessary to give the stress-strain data.

16.12 *PLASTIC: Specify a metal plasticity model.

This option is used to specify the plastic part of the material model for elastic-plastic materials that use the Mises or Hill yield surface.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Classical metal plasticity," Section 20.2.1 of the Abaqus Analysis User's Manual
- "Models for metals subjected to cyclic loading," Section 20.2.2 of the Abaqus Analysis User's Manual
- "Johnson-Cook plasticity," Section 20.2.7 of the Abaqus Analysis User's Manual
- "Permanent set in rubberlike materials," Section 20.7.1 of the Abaqus Analysis User's Manual
- "UHARD," Section 1.1.31 of the Abaqus User Subroutines Reference Manual
- "VUHARD," Section 1.2.14 of the Abaqus User Subroutines Reference Manual

Optional parameters:

HARDENING

Set HARDENING=ISOTROPIC (default) to specify isotropic hardening.
Set HARDENING=KINEMATIC to specify linear kinematic hardening.
Set HARDENING=COMBINED to specify nonlinear isotropic/kinematic hardening.
Set HARDENING=JOHNSON COOK to specify Johnson-Cook hardening.
Set HARDENING=USER to define isotropic hardening in user subroutine **UHARD** in an Abaqus/Standard analysis or user subroutine **VUHARD** in an Abaqus/Explicit analysis.

SCALESTRESS

This parameter cannot be used with the HARDENING parameter.
Set this parameter equal to the factor by which you want the yield stress to be scaled.

Optional parameter for use with HARDENING=ISOTROPIC or HARDENING=COMBINED:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of hardening behavior, in addition to temperature and possibly strain range. If this parameter is omitted, the hardening behavior does not depend on field variables. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Optional parameter for use with HARDENING=ISOTROPIC:

RATE

Set this parameter equal to the equivalent plastic strain rate, $\dot{\bar{\varepsilon}}^{pl}$, for which this stress-strain curve applies.

Optional parameters for use with HARDENING=COMBINED:

DATA TYPE

Set DATA TYPE=HALF CYCLE (default) to specify stress versus plastic strain values of the first half-cycle for calibrating the kinematic hardening parameters.

Set DATA TYPE=PARAMETERS to specify the calibrated kinematic hardening material parameters directly.

Set DATA TYPE=STABILIZED to specify stress versus plastic strain values of a stabilized cycle for calibrating the kinematic hardening parameters.

NUMBER BACKSTRESSES

Set this parameter equal to the number of backstresses. The default number of backstresses is 1, and the maximum allowed is 10.

Optional parameter for use with HARDENING=USER:

PROPERTIES

Set this parameter equal to the number of property values needed as data in user subroutine **UHARD** in Abaqus/Standard analyses and user subroutine **VUHARD** in Abaqus/Explicit analyses. The default is PROPERTIES=0.

Data lines for HARDENING=ISOTROPIC or HARDENING=COMBINED with DATA TYPE=HALF CYCLE:

First line:

1. Yield stress.
2. Plastic strain.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on temperature and other predefined field variables.

Data lines for HARDENING=COMBINED with DATA TYPE=STABILIZED:
First line:

1. Yield stress.
2. Plastic strain.
3. Strain range.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress on plastic strain and, if needed, on strain range, temperature, and other predefined field variables.

Data lines for HARDENING=COMBINED with DATA TYPE=PARAMETERS:
First line:

1. Yield stress at zero plastic strain.
2. Kinematic hardening parameter, C_1.
3. Kinematic hardening parameter, γ_1.
4. Etc., specify kinematic hardening parameters C_k and γ_k for each backstress k. (Only needed if the NUMBER BACKSTRESSES parameter has a value greater than 1.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Etc., up to eight entries per line.

Subsequent lines (only needed if the number of entries is greater than eight):

1. Etc., up to eight entries per line.

Repeat this set of data lines as often as necessary to define the dependence of yield stress and kinematic hardening parameters C_k and γ_k on temperature and other predefined field variables.

Data lines for HARDENING=KINEMATIC:
First line:

1. Yield stress.
2. Plastic strain.

3. Temperature, if temperature dependent.

Repeat this data line a maximum of two times to define linear kinematic hardening independent of temperature. Repeat this set of data lines as often as necessary to define a variation of the linear kinematic hardening modulus with respect to temperature.

Data line for HARDENING=JOHNSON COOK:

First (and only) line:

1. A.
2. B.
3. n.
4. m.
5. Melting temperature, θ_{melt}.
6. Transition temperature, $\theta_{\text{transition}}$.

Data lines for HARDENING=USER with PROPERTIES:

First line:

1. Give the hardening properties, eight per line.

Repeat this data line as often as necessary to define all hardening properties.

16.13 *PLASTIC AXIAL: Define plastic axial force for frame elements.

This option can be used only in conjunction with the *FRAME SECTION option. It describes the axial force in a frame element as a function of the axial plastic displacement.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Frame section behavior," Section 26.4.2 of the Abaqus Analysis User's Manual
- *FRAME SECTION

There are no parameters associated with this option.

Data lines to define the variation of axial yield force with plastic displacement:

First line:

1. Axial yield force.
2. Plastic axial displacement.

Repeat this data line as often as necessary to define the relationship between the axial force and the axial plastic displacement. At least three pairs of data are required.

16.14 *PLASTIC M1: Define the first plastic bending moment behavior for frame elements.

This option can be used only in conjunction with the *FRAME SECTION option. It describes the bending moment in a frame element as a function of the plastic rotation about the first cross-section direction.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Frame section behavior," Section 26.4.2 of the Abaqus Analysis User's Manual
- *FRAME SECTION

There are no parameters associated with this option.

Data lines to define the variation of yield moment M_1^0 with plastic rotation:

First line:

1. Bending yield moment M_1^0 about the first cross-section direction.
2. Plastic rotation about the first cross-section direction.

Repeat this data line as often as necessary to define the relationship between the bending moment and the plastic rotation about the first cross-section axis. At least three pairs of data are required.

16.15 *PLASTIC M2: Define the second plastic bending moment behavior for frame elements.

This option can be used only in conjunction with the *FRAME SECTION option and is available only for FRAME3D elements. It describes the bending moment in a frame element as a function of the plastic rotation about the second cross-section direction.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Frame section behavior," Section 26.4.2 of the Abaqus Analysis User's Manual
- *FRAME SECTION

There are no parameters associated with this option.

Data lines to define the variation of yield moment M_2^0 with plastic rotation:

First line:

1. Bending yield moment M_2^0 about the second cross-section direction.
2. Plastic rotation about the second cross-section direction.

Repeat this data line as often as necessary to define the relationship between the bending moment and the plastic rotation about the second cross-section axis. At least three pairs of data are required.

16.16 *PLASTIC TORQUE: Define the plastic torsional moment behavior for frame elements.

This option can be used only in conjunction with the *FRAME SECTION option and is available only for FRAME3D elements. It describes the torsional moment in a frame element as a function of the plastic rotation about the element's axis.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Frame section behavior," Section 26.4.2 of the Abaqus Analysis User's Manual
- *FRAME SECTION

There are no parameters associated with this option.

Data lines to define the variation of torsional yield moment with plastic rotation:

First line:

1. Torsional yield moment, T^0.
2. Plastic rotation about the element's axis.

Repeat this data line as often as necessary to define the relationship between the torsional moment and the plastic rotation. At least three pairs of data are required.

16.17 *POROUS BULK MODULI: Define bulk moduli for soils and rocks.

This option is used to define the bulk moduli of solid grains and a permeating fluid such that their compressibility can be considered in the analysis of a porous medium. The *POROUS BULK MODULI option cannot be used with the porous metal plasticity material model.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Porous bulk moduli," Section 23.7.3 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data lines to define bulk moduli:

First line:

1. Bulk modulus of solid grains. (Units of FL^{-2}.) If this entry is blank or zero, this value is assumed to be infinite.
2. Bulk modulus of permeating fluid. (Units of FL^{-2}.) If this entry is blank or zero, this value is assumed to be infinite.
3. Temperature.

Repeat this data line as often as necessary to define the dependence of the bulk moduli on temperature.

16.18 *POROUS ELASTIC: Specify elastic material properties for porous materials.

This option is used to define the elastic parameters for porous materials.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Elastic behavior of porous materials," Section 19.3.1 of the Abaqus Analysis User's Manual

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the parameters, in addition to temperature. If this parameter is omitted, it is assumed that the parameters depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

SHEAR

Set SHEAR=G to define the deviatoric behavior with a constant shear modulus G. Set SHEAR=POISSON (default) to compute the instantaneous shear modulus from the bulk modulus and Poisson's ratio. The Poisson's ratio should be given on the data lines.

Data lines to define the deviatoric behavior with a constant shear modulus *G*:

First line:

1. Value of the logarithmic bulk modulus, κ. (Dimensionless.)
2. Value of the shear modulus, G.
3. Value of the elastic tensile limit, p_t^{el}. (This value cannot be negative.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters κ, G, and p_t^{el} on temperature and field variables.

Data lines to define the instantaneous shear modulus from the bulk modulus and Poisson's ratio:

First line:

1. Value of the logarithmic bulk modulus, κ. (Dimensionless.)
2. Value of Poisson's ratio, ν.
3. Value of the elastic tensile limit, p_t^{el}. (This value cannot be negative.)
4. Temperature, θ.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters κ, ν, and p_t^{el} on temperature and field variables.

16.19 *POROUS FAILURE CRITERIA: Define porous material failure criteria for a *POROUS METAL PLASTICITY model.

This option is used to specify the material failure criteria in a porous metal.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- *POROUS METAL PLASTICITY
- "Porous metal plasticity," Section 20.2.9 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data line to define material failure criteria:

First (and only) line:

1. Void volume fraction at total failure, $f_F > 0$. The default is 1.
2. Critical void volume fraction (threshold of rapid loss of stress carrying capacity), $f_c \geq 0$. The default is f_F.

16.20 *POROUS METAL PLASTICITY: Specify a porous metal plasticity model.

This option is used to specify the porous part of the porous metal plasticity model.

The *POROUS METAL PLASTICITY option can be used in conjunction with the *VOID NUCLEATION option to define the nucleation of voids. In an Abaqus/Explicit analysis it can also be used in conjunction with the *POROUS FAILURE CRITERIA option to specify the material failure criteria.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Porous metal plasticity," Section 20.2.9 of the Abaqus Analysis User's Manual
- *VOID NUCLEATION
- *POROUS FAILURE CRITERIA

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, included in the definition of the q_1, q_2, and q_3 parameters. If this parameter is omitted, q_1, q_2, and q_3 may depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

RELATIVE DENSITY

Set this parameter equal to r_0, the initial relative density of the material. If this parameter is omitted, the initial relative density is interpolated from the values given in the *INITIAL CONDITIONS, TYPE=RELATIVE DENSITY option.

Data lines to define porous metal plasticity:

First line:

1. q_1.
2. q_2.
3. q_3.
4. Temperature.
5. First field variable.

6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of q_1, q_2, and q_3 on temperature and other predefined field variables.

16.21 *POST OUTPUT: Postprocess for output from the restart file.

This option can be used only for postprocessing to recover additional printed (`.dat`), output database (`.odb`), and results file (`.fil`) output from the restart file of a previous analysis.

Product: Abaqus/Standard

Type: History data

Level: Model

References:

- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual
- "Restarting an analysis," Section 9.1.1 of the Abaqus Analysis User's Manual

Required parameter:

STEP

Set this parameter equal to the step number from which output is required.

Optional parameters:

CYCLE

This parameter applies only to postprocessing a low-cycle fatigue analysis (see "Low-cycle fatigue analysis using the direct cyclic approach," Section 6.2.7 of the Abaqus Analysis User's Manual).

Set this parameter equal to the cycle number in a low-cycle fatigue analysis from which additional output is being requested.

ITERATION

This parameter applies only to postprocessing a direct cyclic analysis (see "Direct cyclic analysis," Section 6.2.6 of the Abaqus Analysis User's Manual).

Set this parameter equal to the iteration number in a direct cyclic analysis from which additional output is being requested.

Data lines to request output from specified increments if both the ITERATION and the CYCLE parameters are omitted:

First line:

1. First increment number at which additional output is being requested.
2. Second increment number at which additional output is being requested.
3. Etc., up to eight increments per line.

Repeat this data line as often as necessary to define the increments at which output is required.

To recover additional output from a previous direct cyclic analysis when the ITERATION parameter is included:

No data lines are needed when the ITERATION parameter is specified.

To recover additional output from a previous low-cycle fatigue analysis when the CYCLE parameter is included:

No data lines are needed when the CYCLE parameter is specified.

16.22 *POTENTIAL: Define an anisotropic yield/creep model.

This option is used to define stress ratios for anisotropic yield and creep behavior. It can be used only in conjunction with material models defined by the *CREEP option, the *PLASTIC option (HARDENING=ISOTROPIC, KINEMATIC, or COMBINED; the *POTENTIAL option can be used in conjunction with COMBINED hardening only in Abaqus/Explicit), and/or the *VISCOUS option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Classical metal plasticity," Section 20.2.1 of the Abaqus Analysis User's Manual
- "Models for metals subjected to cyclic loading," Section 20.2.2 of the Abaqus Analysis User's Manual
- "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual
- "Anisotropic yield/creep," Section 20.2.6 of the Abaqus Analysis User's Manual
- "Two-layer viscoplasticity," Section 20.2.11 of the Abaqus Analysis User's Manual

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of R_{ij} in addition to temperature. If this parameter is omitted, it is assumed that the anisotropy ratios are constant or depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines used to define stress ratios:

First line:

1. R_{11}.
2. R_{22}.
3. R_{33}.
4. R_{12}.
5. R_{13}.
6. R_{23}.

7. Temperature.
8. First field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than one):

1. Second field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of R_{ij} on temperature and other field variables.

16.23 *PREPRINT: Select printout for the analysis input file processor.

This option is used to select the printout that will be obtained from the analysis input file processor.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Job module

Reference:

- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

CONTACT

This parameter applies only to Abaqus/Standard analyses.

Set CONTACT=YES to print detailed information about the contact constraints generated by the contact pair definition data. Set CONTACT=NO (default) to suppress this printout.

ECHO

Set ECHO=YES to print an echo of the input data. Set ECHO=NO (default) to suppress this printout.

HISTORY

Set HISTORY=YES to print history data. Set HISTORY=NO (default) to suppress this printout.

MODEL

Set MODEL=YES to print model definition data. Set MODEL=NO (default) to suppress this printout.

In Abaqus/Explicit, setting MODEL=YES automatically sets MASS PROPERTY=YES.

PARSUBSTITUTION

Set PARSUBSTITUTION=YES to print the modified version of the original input file that is free of input model parametrization. Set PARSUBSTITUTION=NO (default) to suppress this printout.

PARVALUES

Set PARVALUES=YES to print a modified version of the original input file showing the parameters used for model parametrization and their values. Set PARVALUES=NO (default) to suppress this printout.

MASS PROPERTY

This parameter applies only to Abaqus/Explicit analyses.

Set MASS PROPERTY=YES to print a mass property table, which includes the original mass, initial mass scaling, non-structural mass, center of mass, and rotary inertia of each user-defined element set. Set MASS PROPERTY=NO (default) to suppress this printout.

If MODEL=YES, the mass property table will be printed, even if MASS PROPERTY= NO.

There are no data lines associated with this option.

16.24 *PRESSURE PENETRATION: Specify pressure penetration loads with surface-based contact.

This option is used to prescribe pressure penetration loading simulated with surface-based contact.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

Reference:

- "Pressure penetration loading," Section 33.1.7 of the Abaqus Analysis User's Manual

Required parameters:

MASTER

Set this parameter equal to the name of the master surface of the contact pair used in the pressure penetration analysis.

SLAVE

Set this parameter equal to the name of the slave surface of the contact pair used in the pressure penetration analysis.

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that defines the variation of the fluid pressure during the step. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or ramped up linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option (see "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

OP

Set OP=MOD (default) for existing pressure penetration loads to remain, with this option modifying existing pressure penetration loads or defining additional pressure penetration loads.

Set OP=NEW if all existing pressure penetration loads applied to the model should be removed. New pressure penetration loads can be defined.

PENETRATION TIME

Set this parameter equal to a time period over which the fluid pressure on newly penetrated contact surface segments is ramped up to the current magnitude. The default penetration time period is chosen to be 0.001 of the current step time. This parameter will be neglected in a linear perturbation analysis, in which case the current fluid pressure will be applied immediately once the pressure penetration criterion is satisfied.

Optional, mutually exclusive parameters for matrix generation and steady-state dynamics analysis (direct or modal):

IMAGINARY

Include this parameter to define the imaginary (out-of-phase) part of the loading.

REAL

Include this parameter (default) to define the real (in-phase) part of the loading.

Data lines to define the pressure penetration loads:

First line:

1. A node on the slave surface that is exposed to the fluid.
2. A node on the master surface that is exposed to the fluid. This field should be left blank if the master surface is an analytical rigid surface.
3. Fluid pressure magnitude.
4. Enter the critical contact pressure below which fluid penetration starts to occur. The higher this value, the easier the fluid penetrates. The default is zero, in which case fluid penetration occurs only if contact is lost.

Repeat this data line as often as necessary to define fluid penetrations from different locations on the surface, possibly with different fluid pressure magnitudes. Alternatively, the *PRESSURE PENETRATION option can be repeated (if, for example, a different amplitude reference is needed).

16.25 *PRESSURE STRESS: Specify equivalent pressure stress as a predefined field for a mass diffusion analysis.

This option can be used only in a *MASS DIFFUSION analysis to specify pressure as a predefined field. The user defines equivalent pressure stresses at the nodes, and Abaqus/Standard interpolates the pressure to the material points.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Mass diffusion analysis," Section 6.9.1 of the Abaqus Analysis User's Manual
- "Predefined fields," Section 30.6.1 of the Abaqus Analysis User's Manual

Optional parameters for using the data line format:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that gives the time variation of the pressure throughout the step. If the AMPLITUDE parameter is omitted, the reference magnitude is applied either immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option (see "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

OP

Set OP=MOD (default) for existing *PRESSURE STRESS values to remain, with this option modifying existing values or defining additional values.

Set OP=NEW if all existing *PRESSURE STRESS values should be removed. New pressure stress values can be defined.

For a general analysis step, a pressure that is removed via OP=NEW is reset to the value given on the *INITIAL CONDITIONS option or to zero if no initial pressure was defined. For a linear perturbation step, a pressure that is removed via OP=NEW is always reset to zero. If pressures are being returned to their initial condition values, the AMPLITUDE parameter described above does not apply. Rather, the AMPLITUDE parameter given on the *STEP option governs the behavior. If

pressures are being reset to new values (not to initial conditions) via OP=NEW, the AMPLITUDE parameter described above applies.

Required parameter for reading equivalent pressure stresses from the results file:

FILE

Set this parameter equal to the name of the results file (including the optional **.fil** extension) from which the data are read. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names.

Optional parameters for reading equivalent pressure stresses from the results file:

BSTEP

Set this parameter equal to the step number (of the analysis whose results file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus/Standard will begin reading pressure data from the first step available on the results file.

BINC

Set this parameter equal to the increment number (of the analysis whose results file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus/Standard will begin reading pressure data from the first increment available (excluding any zero increments) for step BSTEP on the results file.

ESTEP

Set this parameter equal to the step number (of the analysis whose results file is being used as input to this option) that ends the history data to be read. If no value is supplied, ESTEP is taken as equal to BSTEP.

EINC

Set this parameter equal to the increment number (of the analysis whose results file is being used as input to this option) that ends the history data to be read. If no value is supplied, EINC is taken as the last available increment of step ESTEP on the results file.

Required parameter for defining data in user subroutine UPRESS:

USER

Include this parameter to indicate that user subroutine **UPRESS** will be used to define equivalent pressure stress values. **UPRESS** will be called for each node given on the data lines. If values are also given on the data lines, these values will be ignored. If a results file has been specified in addition to user subroutine **UPRESS**, values read from the results file will be passed into **UPRESS** for possible modification.

Data lines to define pressures using the data line format:

First line:

1. Node set or node number. If a node set label is given, all nodes in this set must have identical initial pressures.
2. Reference pressure value (positive in compression). If the AMPLITUDE parameter is present, this value will be modified by the AMPLITUDE specification.

Repeat this line as often as necessary to define the pressure at different nodes or node sets.

To read pressures from an Abaqus/Standard results file (FILE):

No data lines are used when pressure data are read from a results file.

Data lines to define equivalent pressure stresses using user subroutine `UPRESS`:

First line:

1. Node set or node number. If a node set label is given, all nodes in this set must have identical initial equivalent pressure stress values.

Repeat this line as often as necessary. **`UPRESS`** will be called for each node listed.

16.26 *PRESTRESS HOLD: Keep rebar prestress constant during initial equilibrium solution.

This option is used within a *STATIC step ("Static stress analysis," Section 6.2.2 of the Abaqus Analysis User's Manual) to keep the stress in some or all of the rebar constant during the initial equilibrium solution.

Product: Abaqus/Standard

Type: History data

Level: Step

Reference:

- "Defining reinforcement," Section 2.2.3 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data lines to hold the prestress constant:

First line:

1. Element set name.
2. Rebar name. The stress in all rebar included in the above element set will be held fixed throughout the step.
3. Etc.

Repeat this data line as often as necessary. Give four pairs of data per line.

16.27 *PRE-TENSION SECTION: Associate a pre-tension node with a pre-tension section.

This option is used to associate a pre-tension node with a pre-tension section.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Load module

Reference:

- "Prescribed assembly loads," Section 30.5.1 of the Abaqus Analysis User's Manual

Required parameter:

NODE

Set this parameter equal to either the pre-tension node number or the name of a node set containing the pre-tension node. If the name of a node set is chosen, the node set must contain exactly one node.

Required, mutually exclusive parameters:

ELEMENT

Set this parameter equal to either the element number of the truss or beam element used to define the pre-tension section or the name of an element set containing the truss or beam element used to define the pre-tension section. If the name of an element set is chosen, the element set must contain exactly one element.

SURFACE

Set this parameter equal to the name of the surface definition (*SURFACE) that defines the pre-tension section (when continuum elements are used).

Data line to define the normal to the section (optional):

First (and only) data line:

1. First component of the normal.
2. Second component of the normal.
3. Third component of the normal.

If the data line is omitted, Abaqus/Standard will compute an average normal to the pre-tension section for continuum elements. For truss or beam elements the default normal points from the first to the last node in the element connectivity.

16.28 *PRINT: Request or suppress output to the message file in an Abaqus/Standard analysis or to the status file in an Abaqus/Explicit analysis.

This option is used to obtain or suppress detailed printout in the message (`.msg`) file in an Abaqus/Standard analysis or in the status (`.sta`) file in an Abaqus/Explicit analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual
- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual

Optional parameters in Abaqus/Standard analyses:

ADAPTIVE MESH

Set ADAPTIVE MESH=YES to request detailed output during adaptive mesh smoothing. The default is ADAPTIVE MESH=NO.

CONTACT

Set CONTACT=YES to request detailed output of points that are contacting or separating in interface and gap problems. This output is useful in difficult contact problems to track the development of the solution during iteration within an increment. The output will be printed for every increment unless FREQUENCY=0. The default is CONTACT=NO.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

MODEL CHANGE

Set MODEL CHANGE=YES to request detailed output of which elements are being removed or reactivated in the step. This output includes the new original coordinates and normals of elements being reactivated strain free in a large-displacement analysis. The default is MODEL CHANGE=NO.

PLASTICITY

Set PLASTICITY=YES to request detailed output of element and integration point numbers for which the plasticity algorithms have failed to converge in the material routines. This output is useful to determine the location in the mesh and the plasticity model for which Abaqus/Standard is encountering material model difficulties. This information may help in identifying modeling problems as well as material parameter specification problems. The default is PLASTICITY=NO.

RESIDUAL

Set RESIDUAL=YES (default) if the output of equilibrium residuals is to be given during the equilibrium iterations. Set RESIDUAL=NO to suppress the output.

SOLVE

Set SOLVE=YES (default) to request information regarding the actual number of equations and the memory requirement in each iteration. Set SOLVE=NO to suppress the output.

Optional parameters in Abaqus/Explicit analyses:

ALLKE

Set ALLKE=YES to request that a column containing the total kinetic energy be printed in the status file. Set ALLKE=NO to suppress this printout. The default is ALLKE=YES.

CRITICAL ELEMENT

Set CRITICAL ELEMENT=YES to request that a column listing the element that has the smallest stable time increment and a column listing the value be printed in the status file. Set CRITICAL ELEMENT=NO to suppress this printout. The default is CRITICAL ELEMENT=YES.

DMASS

Set DMASS=YES to request that a column containing the percent change in total mass of the model due to mass scaling be printed in the status file. Set DMASS=NO to suppress this printout. The default is DMASS=NO unless active mass scaling is present in the step. *FIXED MASS SCALING propagated from a previous step is not considered to be active mass scaling.

ETOTAL

Set ETOTAL=YES to request that a column containing the energy balance of the model be printed in the status file. Set ETOTAL=NO to suppress this printout. The default is ETOTAL=YES.

Other energy variables can also be printed. Please refer to“Total energy output in Abaqus/Standard and Abaqus/Explicit” in “Output to the output database,” Section 4.1.3 of the Abaqus Analysis User’s Manual, for energy variables available in Abaqus/Explicit.

MASS

Set MASS=YES to request that a column containing the total mass of the model be printed in the status file. Set MASS=NO to suppress this printout. The default is MASS=NO.

There are no data lines associated with this option.

16.29 *PSD-DEFINITION: Define a cross-spectral density frequency function for random response loading.

This option is used to define a frequency function for reference in the *CORRELATION option to define the frequency dependence of the random loading in the *RANDOM RESPONSE analysis procedure.

Product: Abaqus/Standard

Type: Model data

Level: Model

References:

- "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Manual
- "UPSD," Section 1.1.43 of the Abaqus User Subroutines Reference Manual
- *CORRELATION
- *RANDOM RESPONSE

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to this frequency function.

Optional parameters:

DB REFERENCE

Set this parameter equal to the reference power value, in $(\text{load units})^2$. This parameter is required when the frequency function is given in decibel units (TYPE=DB).

G

Set this parameter equal to the reference gravity acceleration; for example, 9.81 m/s^2. The default is G=1.0. This parameter can be used only with TYPE=BASE.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

TYPE

Set TYPE=BASE if this frequency function will be used to define a base motion.
Set TYPE=FORCE (default) if this frequency function is given directly in power units.

Set TYPE=DB if this frequency function is defined in decibel units (see below). This option cannot be used with the USER parameter.

USER

Include this parameter if the frequency function is defined in user subroutine **UPSD**. If this parameter is included, no data lines are needed.

Data lines for TYPE=BASE or TYPE=FORCE:

First line:

1. Real part of the frequency function, in units^2 per frequency.
2. Imaginary part of the frequency function, in units^2 per frequency.
3. Frequency, in cycles/time.

Repeat this data line as often as necessary to define the frequency function.

Data lines for TYPE=DB:

First line:

1. Real part of the frequency function, in decibels.
2. Imaginary part of the frequency function, in decibels.
3. Frequency band number (1–15). Up to 15 standard frequency bands can be used.

Repeat this data line as often as necessary to define the frequency function in decibels.

To define the frequency function by a user subroutine (USER parameter included):

No data lines are used with this option if the USER parameter is specified. Instead, user subroutine **UPSD** must be used to define the frequency function.

17. R

17.1 *RADIATE: Specify radiation conditions in heat transfer analyses.

This option is used to apply radiation boundary conditions between a nonconcave surface and a nonreflecting environment in fully coupled thermal-stress analysis. In Abaqus/Standard it is also used for heat transfer and coupled thermal-electrical analyses.

It must be used in conjunction with the *PHYSICAL CONSTANTS option, which is used to define the Stefan-Boltzmann constant.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

Reference:

- "Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the ambient temperature with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option (see "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude given on the data lines is applied throughout the step.

OP

Set OP=MOD (default) for existing *RADIATE definitions to remain, with this option modifying existing radiation conditions or defining additional radiation conditions.

Set OP=NEW if all existing *RADIATE definitions applied to the model should be removed.

REGION TYPE

This parameter applies only to Abaqus/Explicit analyses.

This parameter is relevant only for radiation conditions applied to the boundary of an adaptive mesh domain. If a radiation condition is applied to a surface in the interior of an adaptive mesh domain, the nodes on the surface will move with the material in all directions (they will be nonadaptive). Abaqus/Explicit will create a boundary region automatically on the surface subjected to the defined radiation condition.

Set REGION TYPE=LAGRANGIAN (default) to apply the radiation condition to a Lagrangian boundary region. The edge of a Lagrangian boundary region will follow the material while allowing adaptive meshing along the edge and within the interior of the region.

Set REGION TYPE=SLIDING to apply the radiation condition to a sliding boundary region. The edge of a sliding boundary region will slide over the material. Adaptive meshing will occur along the edge and in the interior of the region. Mesh constraints are typically applied on the edge of a sliding boundary region to fix it spatially.

Set REGION TYPE=EULERIAN to apply the radiation condition to an Eulerian boundary region. This option is used to create a boundary region across which material can flow. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

Data lines to define radiation conditions:

First line:

1. Element number or element set name.
2. Radiation type (see Part VI, “Elements,” of the Abaqus Analysis User’s Manual).
3. Reference ambient temperature value, θ^0. (Units of θ.)
4. Emissivity, ϵ.

Repeat this data line as often as necessary to define radiation conditions.

17.2 *RADIATION FILE: Define results file requests for cavity radiation heat transfer.

This option is used to write cavity radiation variables to the Abaqus/Standard results file.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Unsupported; Abaqus/CAE reads output from the output database file only.

References:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual
- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual

Optional, mutually exclusive parameters (if not specified, output will be provided for all cavities in the model):

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

Optional parameter:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written to the results file at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

Data lines to request results file output:

First line:

1. Give the identifying keys for the variables to be written to the results file for this request. The keys are defined in the “Surface variables” section of “Abaqus/Standard output variable identifiers,” Section 4.2.1 of the Abaqus Analysis User’s Manual.

Repeat this data line as often as necessary to define the surface variables to be written to the results file for the specified cavity, surface, or element set. If this line is omitted, the default variables will be used.

17.3 *RADIATION OUTPUT: Define output database requests for cavity radiation variables.

This option is used to write cavity radiation variables to the output database. It must be used in conjunction with the *OUTPUT option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *OUTPUT

Optional, mutually exclusive parameters (if not specified, output will be provided for all cavities in the model):

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

Optional parameter:

VARIABLE

Set VARIABLE=ALL to indicate that all cavity radiation variables applicable to this procedure and material type should be written to the output database.

If this parameter is omitted, the cavity radiation variables requested for output must be specified on the data lines.

Data lines to request cavity radiation output:

First line:

1. Specify the identifying keys for the variables to be written to the output database. The keys are defined in "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary to define the cavity radiation variables to be written to the output database.

17.4 *RADIATION PRINT: Define print requests for cavity radiation heat transfer.

This option is used to print tabular output of cavity radiation variables (radiation fluxes, viewfactor totals, and facet temperatures).

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual
- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual

Optional, mutually exclusive parameters (if not specified, output will be provided for all cavities in the model):

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

ELSET

Set this parameter equal to the name of the element set for which this output request is being made.

SURFACE

Set this parameter equal to the name of the surface for which this output request is being made.

Optional parameters:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

SUMMARY

Set SUMMARY=YES (default) to obtain a summary of the maximum and minimum values in each column of the table and their locations.

Set SUMMARY=NO to suppress this summary.

TOTALS

Set TOTALS=YES to print the total of each column in the table. This is useful, for example, to sum radiation fluxes over all facets composing a radiation surface. The default is TOTALS=NO.

Data lines to request printed output:

First line:

1. Give the identifying keys for the variables to be printed in a table for this request. The keys are defined in the "Surface variables" section of "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual.

Repeat this data line as often as necessary: each line defines a table (or more than one table if the request is for a cavity made up of more than one surface). If this line is omitted, the default variables will be used.

17.5 *RADIATION SYMMETRY: Define cavity symmetries for radiation heat transfer analysis.

This option must precede the *CYCLIC, *PERIODIC, and/or *REFLECTION options to specify symmetries in cavities used for cavity radiation heat transfer analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual
- *CYCLIC
- *PERIODIC
- *REFLECTION

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the symmetry reference appearing in the *RADIATION VIEWFACTOR option.

There are no data lines associated with this option.

17.6 *RADIATION VIEWFACTOR: Control cavity radiation and viewfactor calculations.

This option is used to control the calculation of viewfactors during a cavity radiation analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

Reference:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

BLOCKING

Set BLOCKING=ALL (default) to specify that full blocking checks be performed in the viewfactor calculations.

Set BLOCKING=NO to specify that no blocking checks be performed in the viewfactor calculations.

Set BLOCKING=PARTIAL to specify that partial blocking checks be performed in the viewfactor calculations. The data lines are then used to specify the potential blocking surfaces.

CAVITY

Set this parameter equal to the name of the cavity for which radiation viewfactor control is being specified. If this parameter is omitted, the specification applies to all cavities defined in the model.

INFINITESIMAL

Set this parameter equal to the facet area ratio above which the infinitesimal-to-finite area approximation is used for viewfactor calculations. The default value is 64.0.

INTEGRATION

Set this parameter equal to the number of Gauss integration points to be used along each edge when the numerical integration of contour integrals is used for viewfactor calculations. One to five integration points are allowed. The default value is three integration points.

LUMPED AREA

Set this parameter equal to the nondimensional distance-square value above which the lumped area approximation is used for viewfactor calculations. The default value is 5.0.

OFF

Include this parameter to switch off cavity radiation effects. Cavity radiation effects can be switched on again by a subsequent use of this option without the OFF parameter. If this parameter is omitted, cavity radiation is active.

RANGE

Set this parameter equal to a distance beyond which viewfactors need not be calculated because surfaces are judged to be too far apart to "see" each other (due to blocking by other surfaces).

REFLECTION

Set REFLECTION=YES (default) to indicate that reflection must be included in the cavity radiation calculations.

Set REFLECTION=NO to indicate that reflection is to be ignored in the cavity radiation calculations. No reflection corresponds to the special case of *black body* radiation (see "Cavity radiation," Section 2.11.4 of the Abaqus Theory Manual).

SYMMETRY

Include this parameter to indicate the existence of radiation symmetries in the model. This parameter must be set equal to the name appearing in the *RADIATION SYMMETRY option where the symmetries are defined. If this parameter is omitted, it is assumed that there are no radiation symmetries in the cavity.

VTOL

Set this parameter equal to the acceptable tolerance for the viewfactor calculations. If this parameter is omitted, the default viewfactor tolerance is 0.05.

Optional parameters (for use with the *MOTION option):

MDISP

Set this parameter equal to the maximum allowable motion of any node in the monitored node set before recalculation of radiation viewfactors. This parameter is relevant only when the *MOTION option appears in conjunction with the *RADIATION VIEWFACTOR option. This parameter must be used with the NSET parameter.

NSET

Set this parameter equal to the name of the node set whose displacements are to be monitored to decide when geometrical changes are significant enough to trigger recalculation of the radiation viewfactors during the step. This parameter is relevant only when the *MOTION option appears in conjunction with the *RADIATION VIEWFACTOR option. This parameter must be used with the MDISP parameter.

Data lines to define blocking surfaces (BLOCKING=PARTIAL):

First line:

1. List of surfaces that provide blocking inside the cavity, up to eight surfaces per line.

Repeat this data line as often as necessary to define partial blocking.

There are no data lines associated with this option if BLOCKING=PARTIAL is not specified.

17.7 *RANDOM RESPONSE: Calculate response to random loading.

This option is used to give the linearized response of a model to random excitation.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Manual
- *CORRELATION
- *PSD-DEFINITION

There are no parameters associated with this option.

Data lines for a random response analysis:

First line:

1. Lower limit of frequency range, in cycles/time.
2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency.
3. Number of points between eigenfrequencies at which the response should be calculated, including the end points, from the lower limit of the frequency range to the first eigenfrequency in the range; in each interval from eigenfrequency to eigenfrequency; and from the highest eigenfrequency in the range to the high limit of the frequency range. If the value given is less than two (or omitted), the default value of 20 points is assumed. Accurate RMS values can be obtained only if enough points are used so that Abaqus/Standard can integrate accurately over the frequency range.
4. Bias parameter. This parameter is useful only if results are requested at four or more frequency points. It is used to bias the results points toward the ends of the intervals so that better resolution is obtained there, since the ends of each interval are the eigenfrequencies where the response amplitudes vary most rapidly. The default bias parameter is 3.0. The bias formula is defined in "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Manual.
5. Frequency scale choice. Enter 1 to use a linear frequency scale. If this field is left blank or is 0, a logarithmic frequency scale is used.

Repeat this data line as often as necessary to define the frequency ranges in which results are required.

17.8 *RATE DEPENDENT: Define a rate-dependent viscoplastic model.

This option can be used only as a suboption of the *PLASTIC, HARDENING=ISOTROPIC option; the *PLASTIC, HARDENING=JOHNSON COOK option; the *DRUCKER PRAGER HARDENING option; or the *CRUSHABLE FOAM HARDENING option to introduce strain rate dependence in the material models.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Rate-dependent yield," Section 20.2.3 of the Abaqus Analysis User's Manual
- "Classical metal plasticity," Section 20.2.1 of the Abaqus Analysis User's Manual
- "Johnson-Cook plasticity," Section 20.2.7 of the Abaqus Analysis User's Manual
- "Extended Drucker-Prager models," Section 20.3.1 of the Abaqus Analysis User's Manual
- "Crushable foam plasticity models," Section 20.3.5 of the Abaqus Analysis User's Manual

Optional parameters:

DEPENDENCIES

In an Abaqus/Explicit analysis this parameter is relevant only for TYPE=POWER LAW or TYPE=YIELD RATIO.

Set this parameter equal to the number of field variable dependencies in the definition of material parameters, in addition to temperature. If this parameter is omitted, it is assumed that the rate-dependent material behavior depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

TYPE

Set TYPE=POWER LAW (default) to specify the Cowper-Symonds overstress power law.

Set TYPE=JOHNSON COOK to specify Johnson-Cook rate dependence (this option cannot be used with the crushable foam plasticity model).

Set TYPE=YIELD RATIO to enter yield stress ratios.

If this option is used in conjunction with the *CRUSHABLE FOAM option, the static hardening relation must be defined on the *CRUSHABLE FOAM HARDENING option.

Data lines to define the overstress power law parameters (TYPE=POWER LAW):

First line:

1. D.
2. n.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the material parameters on temperature and other field variables.

Data line to define the Johnson-Cook rate parameters (TYPE=JOHNSON COOK):

First (and only) line:

1. C.
2. $\dot{\varepsilon}_0$.

Data lines to define yield stress ratios (TYPE=YIELD RATIO):

First line:

1. Yield stress ratio, $R = \bar{\sigma}/\sigma^0$.
2. Equivalent plastic strain rate, $\dot{\bar{\varepsilon}}^{pl}$ (or $\dot{\varepsilon}^{pl}_{axial}|$, the absolute value of the axial plastic strain rate in uniaxial compression, for the crushable foam model).
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the yield stress ratio on equivalent plastic strain and, if needed, on temperature and other predefined field variables.

17.9 *RATIOS: Define anisotropic swelling.

This option is used to specify ratios that define anisotropic swelling. The *RATIOS option can be used only in conjunction with the *MOISTURE SWELLING option or the *SWELLING option, and it should appear immediately after either one.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Moisture swelling," Section 23.7.6 of the Abaqus Analysis User's Manual
- "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual
- *MOISTURE SWELLING
- *SWELLING

Optional parameter when the *RATIOS option is used with the *SWELLING option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the anisotropy ratios in addition to temperature. If this parameter is omitted, it is assumed that the ratios depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define anisotropic swelling ratios:

First line:

1. r_{11}.
2. r_{22}.
3. r_{33}.
4. Temperature. This value and all of the following field variables can be defined only if the *RATIOS option is used with the *SWELLING option.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the anisotropic swelling ratios on temperature and other field variables.

17.10 *REACTION RATE: Define the reaction rate for an ignition and growth equation of state.

This option is used to define the reaction rate for an ignition and growth equation of state. It is required when the *EOS, TYPE=IGNITION AND GROWTH option is used. The *REACTION RATE option should appear immediately after the *EOS or the *GAS SPECIFIC HEAT option.

Products: Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Equation of state," Section 22.2.1 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data lines to define the reaction rate:

First line:

1. Initial pressure, *I*. (Units of T^{-1}.)
2. Product co-volume, *a*. (Dimensionless.)
3. Exponent on the unreacted fraction (ignition term), *b*. (Dimensionless.)
4. Exponent (ignition term), *x*. (Dimensionless.)
5. First burn rate coefficient, G_1. (Units of T^{-1}.)
6. Exponent on the unreacted fraction (growth term), *c*. (Dimensionless.)
7. Exponent on the reacted fraction (growth term), *d*. (Dimensionless.)
8. Pressure exponent (growth term), *y*. (Dimensionless.)

Second line:

1. Second burn rate coefficient, G_2. (Units of T^{-1}.)
2. Exponent on the unreacted fraction (completion term), *e*. (Dimensionless.)
3. Exponent on the reacted fraction (completion term), *g*. (Dimensionless.)
4. Pressure exponent (completion term), *z*. (Dimensionless.)
5. Initial reacted fraction, F_{ig}^{max}. (Dimensionless.)
6. Maximum reacted fraction for the growth term, F_{G1}^{max}. (Dimensionless.)
7. Minimum reacted fraction for the completion term, F_{G2}^{min}. (Dimensionless.)

17.11 *REBAR: Define rebar as an element property.

This option is used as an alternative method to define rebar as an element property in shells, membranes, and solid (continuum) elements. It must be used to define rebar in beams in Abaqus/Standard analyses. The preferred option for defining rebar in shells, membranes, and surface elements is the *REBAR LAYER option, which must be used in conjunction with the *SHELL SECTION, the *MEMBRANE SECTION, or the *SURFACE SECTION options. The preferred method for defining rebar in solids is to embed reinforced surface or membrane elements in "host" solid elements using the *EMBEDDED ELEMENT option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Defining rebar as an element property," Section 2.2.4 of the Abaqus Analysis User's Manual
- "Defining reinforcement," Section 2.2.3 of the Abaqus Analysis User's Manual

Required parameters:

ELEMENT

Set ELEMENT=BEAM to define rebar in beam elements in an Abaqus/Standard analysis.

Set ELEMENT=SHELL to define rebar in three-dimensional shell elements. Rebar cannot be used with triangular shell elements.

Set ELEMENT=AXISHELL to define rebar in axisymmetric shell elements.

Set ELEMENT=MEMBRANE to define rebar in three-dimensional membrane elements. Rebar cannot be used with triangular membrane elements.

Set ELEMENT=AXIMEMBRANE to define rebar in axisymmetric membrane elements in an Abaqus/Standard analysis.

Set ELEMENT=CONTINUUM to define rebar in continuum (solid) elements. Rebar cannot be used with any plane triangular, triangular prism, tetrahedral, or infinite elements.

MATERIAL

Set this parameter equal to the name of the material of which these rebar are made.

NAME

Set this parameter equal to a label that will be used to refer to this rebar set. This label can be used in defining rebar prestress and output requests. Each layer of rebar must be assigned a separate name in a particular element or element set.

Optional parameters:

GEOMETRY

This parameter is not meaningful for rebar in beams, axisymmetric shells, or axisymmetric membranes, or for single rebar in continuum elements.

Set GEOMETRY=ISOPARAMETRIC (default) to indicate that the layer of rebar is parallel to a direction of the element local (isoparametric) coordinate system.

Set GEOMETRY=SKEW to indicate that the rebar layer is in a skew direction with respect to the element faces.

ISODIRECTION

Set this parameter equal to the isoparametric direction from which the rebar angle output will be measured. The default is 1.

ORIENTATION

This parameter is meaningful only for skew rebar in shell and membrane elements. Set this parameter equal to the name of an orientation definition that defines the angular orientation of the rebar. This parameter is not permitted with axisymmetric shell and axisymmetric membrane elements.

SINGLE

This parameter is meaningful only for continuum elements. Include this parameter if a single rebar is being defined by each data line. If this parameter is omitted, each line defines a layer of uniformly spaced rebar in the element isoparametric space.

Data lines to define rebar in beam elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of the rebar.
3. Distance X_1 (see Figure 17.11–1).
4. Distance X_2.

Repeat this data line as often as necessary. Each line defines a single rebar.

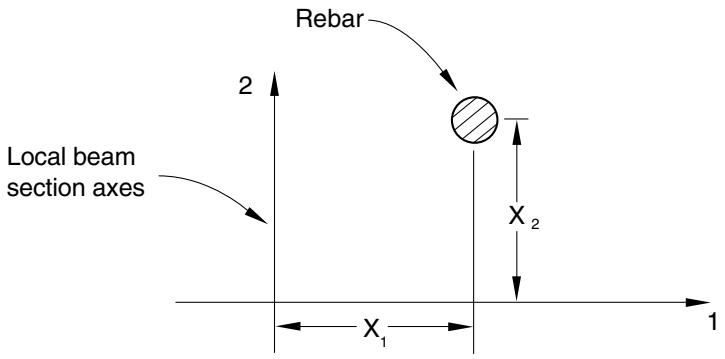


Figure 17.11–1 Rebar location in a beam section.

Data lines to define isoparametric rebar in three-dimensional shell elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of the rebar in the plane of the shell. The default is 1.0.
4. Position of the rebar in the shell section thickness direction. This value is given as the distance of the rebar from the middle surface of the shell, positive in the direction of the positive normal to the shell. This value is modified if the NODAL THICKNESS parameter is included with the *SHELL SECTION option of the underlying shell element.
5. Edge number to which the rebar are parallel in the element's local (isoparametric) coordinate system. See Figure 17.11–2.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define isoparametric rebar in three-dimensional membrane elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of the rebar in the plane of the membrane. The default is 1.0.
4. Edge number to which the rebar are parallel in the element's local (isoparametric) coordinate system. See Figure 17.11–2.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

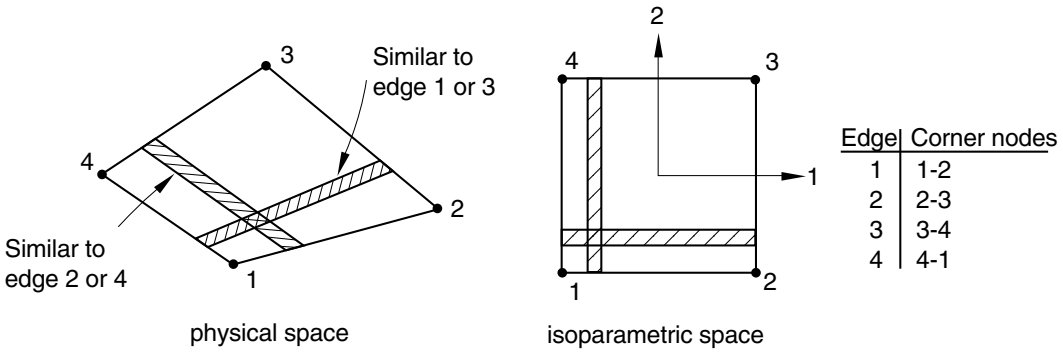


Figure 17.11–2 "Isoparametric" rebar in a three-dimensional shell or membrane.

Data lines to define skew rebar in three-dimensional shell elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar in the plane of the shell. The default is 1.0.
4. Position of the rebar in the shell section thickness direction. This value is given as the distance of the rebar from the middle surface of the shell, positive in the direction of the positive normal to the shell. This value is modified if the NODAL THICKNESS parameter is included with the *SHELL SECTION option of the underlying shell element.
5. Angular orientation of rebar (in degrees) between the positive local 1-direction and the rebar. The optional ORIENTATION parameter given on the *SHELL SECTION option should have no influence on the rebar angular orientation.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define skew rebar in three-dimensional membrane elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar in the plane of the membrane. The default is 1.0.
4. Angular orientation of rebar (in degrees) between the positive local 1-direction and the rebar. The optional ORIENTATION parameter given on the *MEMBRANE SECTION option should have no influence on the rebar angular orientation.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define rebar in axisymmetric shell elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar in this rebar layer. The default is 1.0.
4. Position of the rebar in the shell section thickness direction. This value is given as the distance of the rebar from the middle surface of the shell, positive in the direction of the positive normal to the shell. This value is modified if the NODAL THICKNESS parameter is included with the *SHELL SECTION option of the underlying shell element.
5. Angular orientation of rebar from the meridional plane in degrees (0° is meridional, 90° is circumferential). Positive rotation is about the positive normal to the shell.
6. Radial position at which the spacing of the rebar is measured. If this entry is nonzero, it is assumed that the rebar spacing varies linearly with radial position. If this entry is zero or blank, the rebar spacing does not vary with position. This entry has no meaning for circumferential rebar.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define rebar in axisymmetric membrane elements:

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar in this rebar layer. The default is 1.0.
4. Angular orientation of rebar from the meridional plane in degrees (0° is meridional, 90° is circumferential). Positive rotation is about the positive normal to the membrane.
5. Radial position at which the spacing of the rebar is measured. If this entry is nonzero, it is assumed that the rebar spacing varies linearly with radial position. If this entry is zero or blank, the rebar spacing does not vary with position. This entry has no meaning for circumferential rebar.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define a layer of uniformly spaced rebar in continuum elements (SINGLE parameter omitted) when the layer is parallel to two isoparametric directions in the element's local (isoparametric) coordinate system (GEOMETRY=ISOPARAMETRIC):

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar. The default is 1.0.

4. Orientation of rebar in degrees. See Figure 17.11–3.
5. Fractional distance from the edge given below, f (ratio of the distance between the edge and the rebar to the distance across the element).
6. Edge number from which the rebar are defined. See Figure 17.11–4 or Figure 17.11–7.
7. Isoparametric direction (for three-dimensional elements only).
8. For axisymmetric elements only, the radial position at which the spacing of the rebar is measured. If this entry is nonzero, the rebar spacing is assumed to vary linearly with radial position. If this entry is zero or blank, the rebar spacing does not vary with position.

Repeat this data line as often as necessary. Each line defines a layer of rebar.

Data lines to define a layer of uniformly spaced rebar in continuum elements (SINGLE parameter omitted) when the layer is parallel to only one isoparametric direction in the element's local (isoparametric) coordinate system (GEOMETRY=SKEW):

First line:

1. Element number or name of the element set that contains these rebar.
2. Cross-sectional area of each rebar.
3. Spacing of rebar. The default is 1.0.
4. Orientation of rebar in degrees. See Figure 17.11–3.
5. For axisymmetric elements only, give the radial position at which the spacing of the rebar is measured. If this entry is nonzero, the rebar spacing is assumed to vary linearly with radial position. If this entry is zero or blank, the rebar spacing does not vary with position.
6. Isoparametric direction (for three-dimensional elements only).

Second line:

1. Fractional distance along edge 1, f_1 (see Figure 17.11–5).
2. Fractional distance along edge 2, f_2.
3. Fractional distance along edge 3, f_3.
4. Fractional distance along edge 4, f_4.

Only the two values corresponding to the two edges that the rebar intersects can be nonzero.

Repeat the first and second data lines as often as necessary. Each pair of lines defines a layer of rebar.

Data lines to define a single rebar in continuum elements (SINGLE parameter included):

First line:

1. Element number or name of the element set that contains the rebar.
2. Cross-sectional area of the rebar.
3. Fractional distance, f_1, locating the rebar's position in the element (see Figure 17.11–6).

4. Fractional distance, f_2, locating the rebar's position in the element.
5. Isoparametric direction (for three-dimensional elements only).

In three-dimensional cases the fractional distances f_1, and f_2 are given along the first two edges of the face identified in Figure 17.11–7 for the isoparametric direction chosen.

Repeat this data line as often as necessary. Each line defines a single rebar.

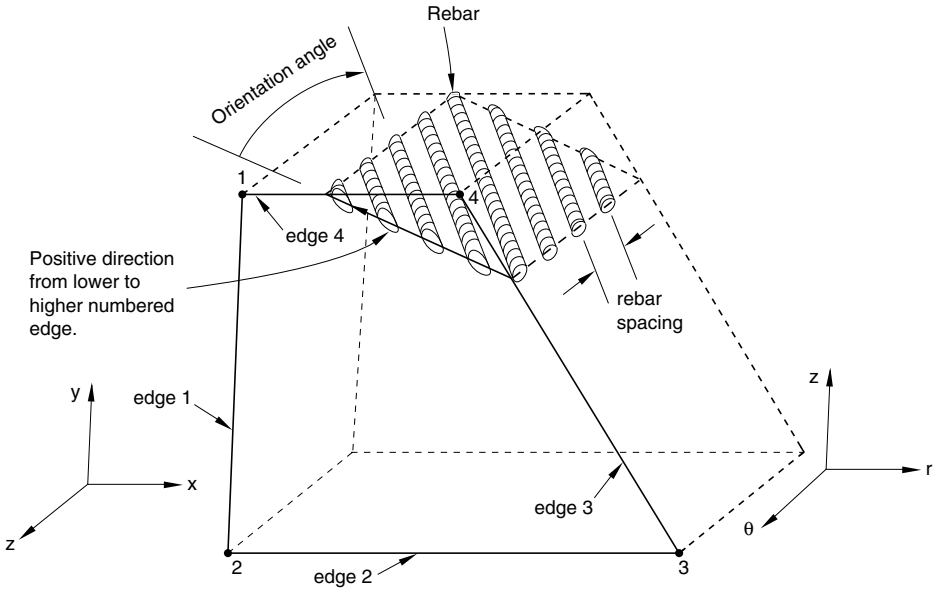


Figure 17.11–3 Orientation of rebar in plane and axisymmetric solid elements.

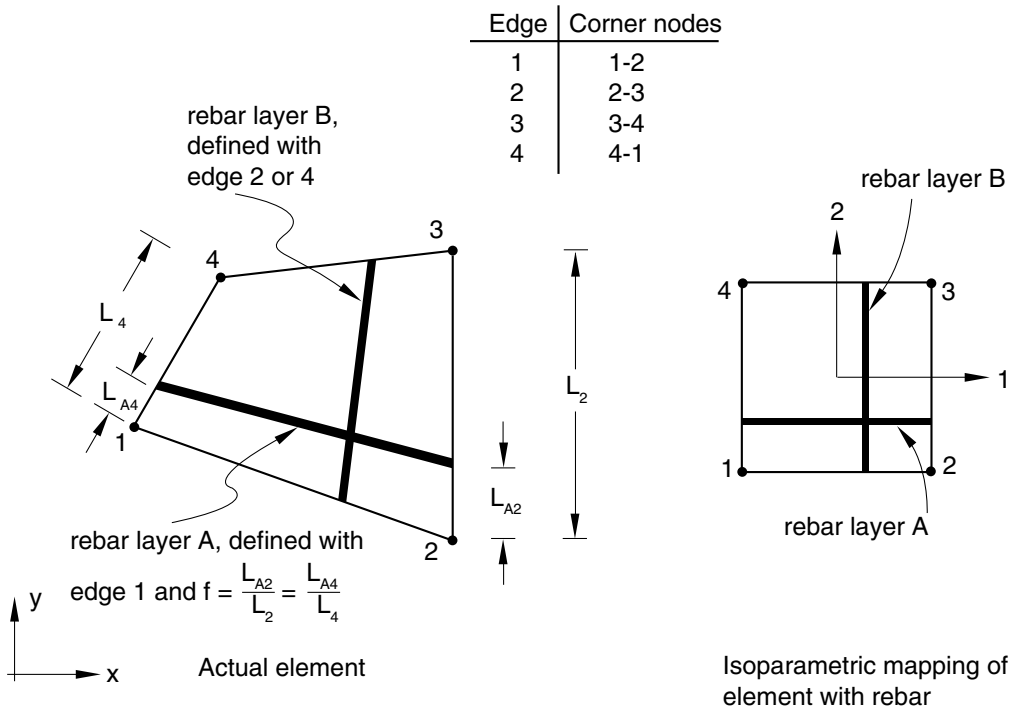


Figure 17.11–4 Rebar layer definition in solid elements with GEOMETRY=ISOPARAMETRIC.

Edge	Corner nodes
1	1-2
2	2-3
3	3-4
4	4-1

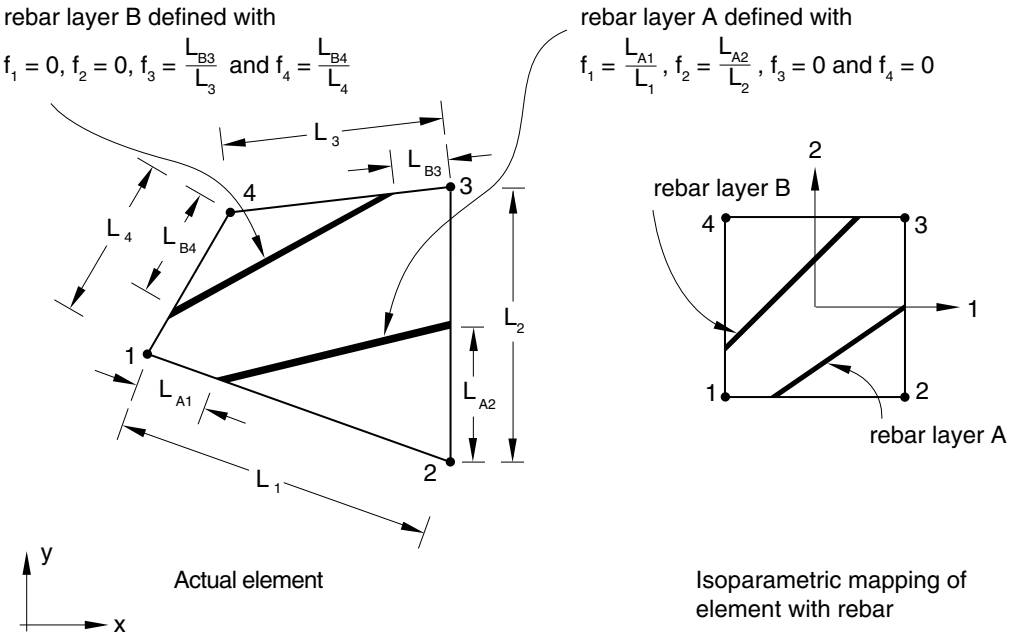


Figure 17.11–5 Rebar layer definition in solid elements with GEOMETRY=SKEW.

Edge	Corner nodes
1	1-2
2	2-3

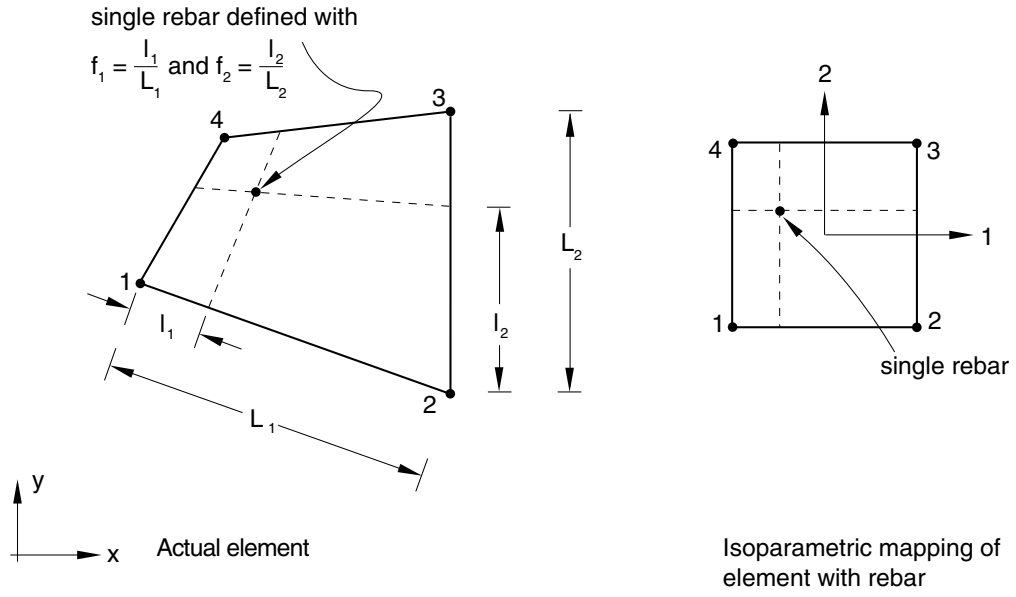
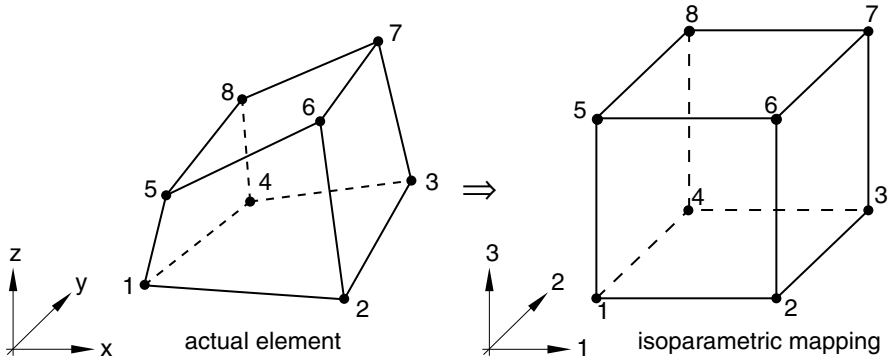


Figure 17.11–6 SINGLE rebar in a solid element.



Isoparametric direction: 1 (parallel to the 1-2 edge of the element and intersecting face 1-4-8-5)

Edge	Corner nodes
1	1-4
2	4-8
3	8-5
4	5-1

Isoparametric direction: 2 (parallel to the 1-4 edge of the element and intersecting face 1-5-6-2)

Edge	Corner nodes
1	1-5
2	5-6
3	6-2
4	2-1

Isoparametric direction: 3 (parallel to the 1-5 edge of the element and intersecting face 1-2-3-4)

Edge	Corner nodes
1	1-2
2	2-3
3	3-4
4	4-1

Figure 17.11–7 Isoparametric direction and edge definitions for three-dimensional elements.

17.12 *REBAR LAYER: Define layers of reinforcement in membrane, shell, surface, and continuum elements.

This option is used to define one or multiple rebar layers in membrane, shell, and surface elements. It must be used in conjunction with the *MEMBRANE SECTION, the *SHELL SECTION, or the *SURFACE SECTION option. Rebar layers in solid (continuum) elements can be defined by embedding a set of surface or membrane elements with rebar layers in a set of host continuum elements through the use of the *EMBEDDED ELEMENT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module; supported only for membrane and shell elements.

References:

- "Defining reinforcement," Section 2.2.3 of the Abaqus Analysis User's Manual
- "Embedded elements," Section 31.4.1 of the Abaqus Analysis User's Manual
- *EMBEDDED ELEMENT
- *MEMBRANE SECTION
- *SHELL SECTION
- *SURFACE SECTION

Optional parameters:

GEOMETRY

Use this parameter to specify the type of rebar geometry.

Set GEOMETRY=CONSTANT (default) if the rebar spacing in the element is constant. The spacing is given as a length measure on the data lines.

Set GEOMETRY=ANGULAR if the rebar spacing increases linearly as a function of the distance measured from the axis of revolution in a cylindrical coordinate system. The spacing is given as an angular spacing in degrees on the data lines. A cylindrical orientation system must be defined if this option is used with three-dimensional membrane, shell, or surface elements.

Set GEOMETRY=LIFT EQUATION if the rebar spacing and orientation within the element is determined by the tire "lift" equation. The rebar parameters are defined with respect to the uncured or "green" tire configuration, and the lift equation maps the rebar parameters to the cured tire configuration. The spacing is given as a length measure on the data lines. A cylindrical orientation

system must be defined if this option is used with three-dimensional membrane, shell, or surface elements.

ORIENTATION

This parameter is meaningful only for rebar in general shell, membrane, and surface elements. Set this parameter equal to the name of an orientation definition that defines the angular orientation of the rebar on the data lines. If this parameter is omitted, the angular orientation of rebar on the data lines is specified relative to the default projected local surface directions. This parameter is not permitted with axisymmetric shell, axisymmetric membrane, and axisymmetric surface elements.

Data lines to define rebar layers:

First line:

1. A name that will be used to identify this rebar layer. This name can be used in defining rebar prestress and output requests. Each layer of rebar must be given a separate name in a particular element or element set.
2. Cross-sectional area of the rebar.
3. Spacing of rebar in the plane of the membrane, shell, or surface element. The value is given as a length measure unless the GEOMETRY=ANGULAR parameter is specified, in which case the value should be given in terms of angular spacing in degrees. Specify the spacing in the uncured geometry if GEOMETRY=LIFT EQUATION.
4. Position of the rebar in the shell section thickness direction. This value is given as the distance of the rebar from the middle surface of the shell, positive in the direction of the positive normal to the shell. This value is modified if the NODAL THICKNESS parameter is included with the *SHELL SECTION option for the underlying shell element. This entry has no meaning for rebar in either membrane or surface elements.
5. Name of the material forming this rebar layer.
6. For three-dimensional shell, membrane, and surface elements, specify the angular orientation of rebar (in degrees) between the positive local 1-direction and the rebar. Specify the angle in the uncured geometry if GEOMETRY=LIFT EQUATION.
   The local 1-direction can be defined by using the *ORIENTATION option and setting the ORIENTATION parameter equal to the orientation name. If the ORIENTATION parameter is omitted, the local 1-direction is defined by the default projected local surface directions. The optional ORIENTATION parameter given on the *MEMBRANE SECTION and the *SHELL SECTION options has no influence on the rebar angular orientation.
   For axisymmetric shell, axisymmetric membrane, and axisymmetric surface elements, specify the angular orientation of rebar from the meridional plane in degrees (0° is meridional, 90° is circumferential). Positive rotation is about the positive normal to the shell, membrane, or surface elements.
7. The isoparametric direction from which the rebar angle output will be measured. The default is 1.

8. Extension ratio, e, for rebar defined with GEOMETRY=LIFT EQUATION. In a tire e represents the pre-strain that occurs during the curing process; e =1 means a 100% extension. This entry has no meaning for rebar defined using GEOMETRY=CONSTANT or GEOMETRY=ANGULAR.
9. Radius, r_0, of the rebar defined with GEOMETRY=LIFT EQUATION. The value is the position of the rebar in the uncured geometry, measured with respect to the axis of rotation in a cylindrical coordinate system. This entry has no meaning for rebar defined using GEOMETRY=CONSTANT or GEOMETRY=ANGULAR.

Repeat the data line as often as necessary. Each data line defines a layer of rebar.

17.13 *REFLECTION: Define reflection symmetries for a cavity radiation heat transfer analysis.

This option is used to define a cavity symmetry by reflection through a line or a plane. It can be used only following the *RADIATION SYMMETRY option.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual
- *RADIATION SYMMETRY

Required parameter:

TYPE

Set TYPE=LINE to create a cavity composed of the cavity surface defined in the model and its reflection through a line. See Figure 17.13–1. This option can be used only for two-dimensional cases.

Set TYPE=PLANE to create a cavity composed of the cavity surface defined in the model and its reflection through a plane. See Figure 17.13–2. This option can be used only for three-dimensional cases.

Set TYPE=ZCONST to create a cavity composed by the cavity surface defined in the model and its reflection through a line of constant z-coordinate. See Figure 17.13–3. This option can be used only for axisymmetric cases.

Data line to define reflection of a two-dimensional cavity (TYPE=LINE):

First (and only) line:

1. X-coordinate of point a (see Figure 17.13–1).
2. Y-coordinate of point a.
3. X-coordinate of point b.
4. Y-coordinate of point b.

Data lines to define reflection of a three-dimensional cavity (TYPE=PLANE):

First line:

1. *X*-coordinate of point *a* (see Figure 17.13–2).
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. *X*-coordinate of point *c*.
2. *Y*-coordinate of point *c*.
3. *Z*-coordinate of point *c*.

Data line to define reflection of an axisymmetric cavity (TYPE=ZCONST):

First (and only) line:

1. *z*-coordinate of symmetry line (see Figure 17.13–3).

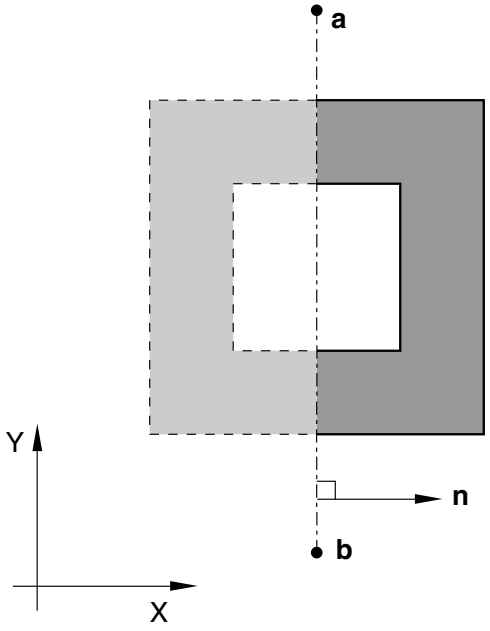


Figure 17.13–1 *REFLECTION, TYPE=LINE option.

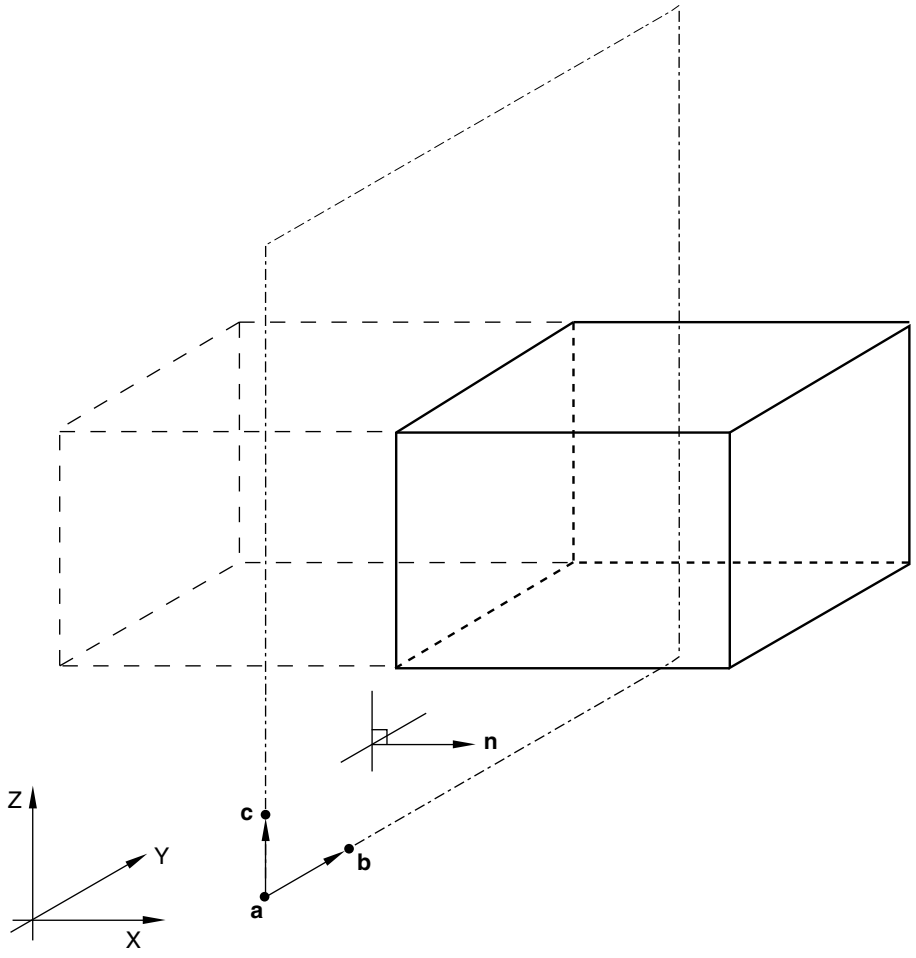


Figure 17.13–2 *REFLECTION, TYPE=PLANE option.

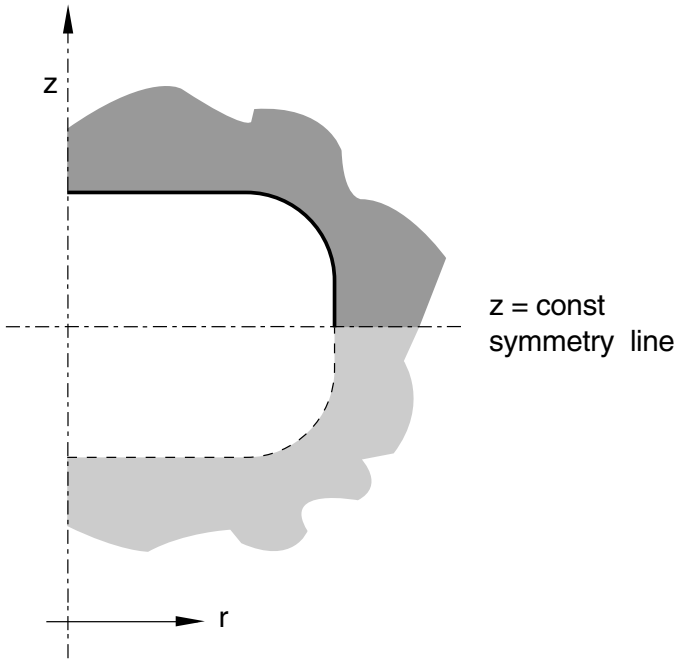


Figure 17.13–3 *REFLECTION, TYPE=ZCONST option.

17.14 *RELEASE: Release rotational degrees of freedom at one or both ends of a beam element.

This option is used to release a rotational degree of freedom or a combination of rotational degrees of freedom at one or both ends of a beam element.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

References:

- "Kinematic coupling constraints," Section 31.2.3 of the Abaqus Analysis User's Manual
- "Element end release," Section 31.5.1 of the Abaqus Analysis User's Manual

Optional parameter:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to prescribe the released degrees of freedom:

First line:

1. Element number or element set label.
2. Element end identifier (S1 or S2).
3. Release combination code at the end of the element (M1, M2, T, M1-M2, M1-T, M2-T, ALLM). See "Element end release," Section 31.5.1 of the Abaqus Analysis User's Manual, for a definition of the release combination codes in Abaqus/Standard. This field can be left blank if none of the rotational degrees of freedom are released at this end of the element.

Repeat this data line as often as necessary to specify the rotational degrees of freedom to be released for different elements and element ends.

17.15 *RESPONSE SPECTRUM: Calculate the response based on user-supplied response spectra.

This option is used to calculate estimates of peak values of displacements and stresses based on user-supplied response spectra (defined using the *SPECTRUM option) and on the natural modes of the system.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Response spectrum analysis," Section 6.3.10 of the Abaqus Analysis User's Manual
- *SPECTRUM

Optional parameters:

COMP

Set COMP=ALGEBRAIC to sum the directional excitation components algebraically. If this parameter is used, the directional excitation components are summed first, followed by the modal summation.

Set COMP=SRSS (default) to use the square root of the sum of the squares. If this parameter is used, the modal summation is performed first, followed by the summation of the directional excitation components.

SUM

Set SUM=ABS (default) to sum the absolute values of the responses in each natural mode.

Set SUM=CQC to use the complete quadratic combination method to sum the responses in each natural mode.

Set SUM=NRL to use the Naval Research Laboratory method.

Set SUM=SRSS to use the square root of the sum of squares summation.

Set SUM=TENP to use the Ten Percent Method.

Data lines for a response spectrum analysis:

First line:

1. Name of the response spectrum to be used.
2. *X*-direction cosine of this direction.
3. *Y*-direction cosine of this direction.

4. *Z*-direction cosine of this direction.
5. Factor multiplying the magnitudes in the response spectrum. Default is 1.0.

Second line (optional):

1. Name of the response spectrum to be used in the second direction.
2. *X*-direction cosine of this direction. This direction must be at a right angle to the direction defined above.
3. *Y*-direction cosine of this direction. This direction must be at a right angle to the direction defined above.
4. *Z*-direction cosine of this direction. This direction must be at a right angle to the direction defined above.
5. Factor multiplying the magnitudes in the response spectrum. Default is 1.0.

Third line (optional):

1. Name of the response spectrum to be used in the third direction.
2. *X*-direction cosine of this direction. This direction must be at right angles to the two directions defined above.
3. *Y*-direction cosine of this direction. This direction must be at right angles to the two directions defined above.
4. *Z*-direction cosine of this direction. This direction must be at right angles to the two directions defined above.
5. Factor multiplying the magnitudes in the response spectrum. Default is 1.0.

17.16 *RESTART: Save and reuse data and analysis results.

WARNING: This option can create a very large amount of data.

This option is used to control the writing and reading of restart data.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Step module for saving restart data; Job module for performing a restart analysis

Using *RESTART in an Abaqus/Standard analysis

Reference:

- "Restarting an analysis," Section 9.1.1 of the Abaqus Analysis User's Manual

At least one of the following parameters is required:

READ

Include this parameter to specify that this analysis is a restart of a previous analysis. The basic model definition data (elements, materials, nodes) cannot be changed at such a restart. However, element sets, node sets, and *AMPLITUDE tables can be added, and history data subsequent to that part of the history already analyzed can be changed.

WRITE

Include this parameter to specify that restart data are to be written during the analysis.

Optional parameters if the READ parameter is used:

CYCLE

If the new analysis is restarted from a previous low-cycle fatigue analysis, set this parameter equal to the cycle number within the specified step of the low-cycle fatigue analysis after which the analysis will resume. Since the new analysis can be restarted only from the end of a loading cycle in the previous low-cycle fatigue analysis, the INC parameter is irrelevant and is ignored if the CYCLE parameter is specified.

If this parameter is omitted, the restart will begin at the end of the last cycle for the specified step.

END STEP

This parameter specifies that the user wishes to terminate the current step in the analysis from which the restart is being made.

This parameter is useful when the user wishes to redefine the loading history, output options, or tolerance controls, etc. If this parameter is included, the data must contain further step definitions to define how the analysis will continue.

If this parameter is omitted, Abaqus/Standard will continue the analysis to complete the current step as it is defined in the run from which the restart is being made.

INC

Set this parameter equal to the increment number within the step specified by the STEP parameter, after which the analysis will resume.

If this parameter is omitted, the restart will begin at the end of the step specified on the STEP parameter.

ITERATION

If the new analysis is restarted from a previous direct cyclic analysis, set this parameter equal to the iteration number within the specified step of the direct cyclic analysis after which the analysis will resume. Since the new analysis can be restarted only from the end of a loading cycle in the previous direct cyclic analysis, the INC parameter is irrelevant and is ignored if the ITERATION parameter is specified.

If this parameter is omitted, the restart will begin at the end of the last iteration for the specified step.

STEP

Set this parameter equal to the step number at which the restart will be made.

If this parameter is omitted, the analysis will restart at the last available step found.

Optional parameters if the WRITE parameter is used:

FREQUENCY

This parameter specifies the increments at which restart information will be written. For example, FREQUENCY=2 will write restart information at increments 2, 4, 6, etc.

For a direct cyclic analysis this parameter specifies the iteration numbers at which restart information will be written; restart information will be written only at the end of an iteration.

For a low-cycle fatigue analysis this parameter specifies the cycle numbers at which restart information will be written; restart information will be written only at the end of a cycle.

When restart is requested in an analysis, restart information is always written at the end of a step. Set FREQUENCY=0 to discontinue the writing of restart data.

The FREQUENCY and NUMBER INTERVAL parameters are mutually exclusive. The default is FREQUENCY=1.

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the *RESTART data are to be written. The value of this parameter must be a positive integer.

The FREQUENCY and NUMBER INTERVAL parameters are mutually exclusive. The default is FREQUENCY=1.

TIME MARKS

Set TIME MARKS=YES (default) to write results at the exact times dictated by the NUMBER INTERVAL parameter.

Set TIME MARKS=NO to write the restart at the increment ending immediately after the time dictated by the NUMBER INTERVAL parameter.

OVERLAY

This parameter specifies that only one increment should be retained per step, thus minimizing the storage space needed.

When the OVERLAY parameter is included, each increment written overlays the previous increment, if any, written for the same step. If this parameter is omitted, data are retained for every increment. In either case the last increment of every step is retained.

There are no data lines associated with this option.

Using *RESTART in an Abaqus/Explicit analysis

Reference:

- "Restarting an analysis," Section 9.1.1 of the Abaqus Analysis User's Manual

At least one of the following parameters is required:

READ

Include this parameter to specify that this analysis is a restart of a previous analysis. The basic model definition data (elements, materials, nodes) cannot be changed at such a restart. However, element sets, node sets, and *AMPLITUDE tables can be added, and history data subsequent to that part of the history already analyzed can be changed.

WRITE

Include this parameter to specify that restart data are to be written during the analysis.

Required parameter if the READ parameter is used:

STEP

Set this parameter equal to the step number at which the restart will be made. If the END STEP and INTERVAL parameters are omitted, the analysis will resume after the last interval available in this step.

Optional parameters if the READ parameter is used:

END STEP

Include this parameter to specify that the current step should be terminated at this point. The INTERVAL parameter is required when this parameter is used.

This parameter is useful when the user wishes to redefine the loading history, contact surface definitions, etc.

If this parameter is omitted, Abaqus/Explicit will continue the analysis from the last available restart interval to complete the current step as it is defined in the run from which the restart is being made.

INTERVAL

Set this parameter equal to the interval number within the step specified by the STEP parameter after which the analysis will resume. The END STEP parameter is required when this parameter is used.

Optional parameters if the WRITE parameter is used:

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which the *RESTART data are to be written. The value of this parameter must be a positive integer. The default is NUMBER INTERVAL=1.

Abaqus/Explicit will always write the restart data at the beginning and end of the step. For example, if NUMBER INTERVAL=10, Abaqus/Explicit will write 11 restart states, consisting of the values at the beginning of the step and the values at 10 evenly spaced intervals throughout the step.

TIME MARKS

Set TIME MARKS=NO (default) to write the restart at the increment ending immediately after the time dictated by the NUMBER INTERVAL parameter.

Set TIME MARKS=YES to write results at the exact times dictated by the NUMBER INTERVAL parameter. TIME MARKS=YES cannot be used when either the FIXED TIME INCREMENTATION or DIRECT USER CONTROL parameter is included on the *DYNAMIC option (“Explicit dynamic analysis,” Section 6.3.3 of the Abaqus Analysis User’s Manual).

OVERLAY

This parameter specifies that only one increment should be retained per step, thus minimizing the storage space needed.

When the OVERLAY parameter is included, each increment written overlays the previous increment, if any, written for the same step. If this parameter is omitted, data are retained for every increment. In either case the last increment of every step is retained.

SINGLE

This parameter specifies that only single precision restart data are to be written when the double precision executable is run.

When the single precision executable is run, this parameter is ignored and only single precision restart data will be written. If this parameter is omitted, double precision restart data will be written when the double precision executable is run.

There are no data lines associated with this option.

17.17 *RETAINED NODAL DOFS: Specify the degrees of freedom that are to be retained as external to a substructure.

This option is used to list degrees of freedom that are to be retained as external degrees of freedom on the substructure. It can be used only in a *SUBSTRUCTURE GENERATE analysis.

Product: Abaqus/Standard

Type: History data

Level: Step

Reference:

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Manual

Optional parameter:

SORTED

Set SORTED=NO to prevent the retained nodes from being sorted. The ordering of the nodes when using a substructure is then the same as the ordering used when specifying the retained nodes. The default is SORTED=YES, the retained nodes are sorted into ascending numerical order.

Data lines to define the retained degrees of freedom:

First line:

1. Node number or node set label.
2. First degree of freedom to be retained.
3. Last degree of freedom to be retained.

If only the node number or node set label is given, all degrees of freedom will be retained.

Repeat this data line as often as necessary to list all degrees of freedom to be retained.

17.18 *RIGID BODY: Define a set of elements as a rigid body and define rigid element properties.

This option is used to bind a set of elements and/or a set of nodes and/or an analytical surface into a rigid body and assign a reference node to the rigid body, which can optionally be declared as an isothermal rigid body for fully coupled thermal-stress analysis. It is also used to specify density, thickness, and offset for rigid elements that are part of a rigid body in an Abaqus/Explicit analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Part module and Interaction module

References:

- "Rigid elements," Section 27.3.1 of the Abaqus Analysis User's Manual
- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Manual
- "Rigid body definition," Section 2.4.1 of the Abaqus Analysis User's Manual

Required parameter:

REF NODE

Set this parameter equal to either the node number of the rigid body reference node or the name of a node set containing the rigid body reference node. If the name of a node set is chosen, the node set must contain exactly one node.

At least one of the following parameters is required:

ANALYTICAL SURFACE

Set this parameter equal to the name of the analytical surface to be assigned to the rigid body.

ELSET

Set this parameter equal to the name of the element set containing the elements to be assigned to the rigid body. An element cannot appear in more than one rigid body.

PIN NSET

Set this parameter equal to the name of a node set containing pin-type nodes to be assigned to the rigid body. This parameter can be used to add nodes to a rigid body or to redefine node types of nodes on elements included in the rigid body by the ELSET parameter. Pin-type nodes have only

their translational degrees of freedom associated with the rigid body. A node cannot appear in more than one rigid body definition.

TIE NSET

Set this parameter equal to the name of a node set containing tie-type nodes to be assigned to the rigid body. This parameter can be used to add nodes to a rigid body or to redefine node types of nodes on elements included in the rigid body by the ELSET parameter. Tie-type nodes have both their translational and rotational degrees of freedom associated with the rigid body. A node cannot appear in more than one rigid body definition.

Optional parameters:

ISOTHERMAL

This parameter is used only for fully coupled thermal-stress analysis.

Set this parameter equal to YES to specify an isothermal rigid body. The default is ISOTHERMAL=NO.

POSITION

Set POSITION=INPUT (default) if the location of the reference node is to be defined by the user.

Set POSITION=CENTER OF MASS if the reference node is to be located at the center of mass of the rigid body.

Optional parameters (use only when the element set specified contains rigid elements):

DENSITY

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the density of all of the rigid elements in the element set.

NODAL THICKNESS

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to indicate that the thickness of the rigid elements should not be read from the data line but should be interpolated from the thickness specified at the nodes with the *NODAL THICKNESS option.

OFFSET

This parameter applies only to Abaqus/Explicit analyses.

Include this parameter to define the distance (as a fraction of the rigid element's thickness) from the element's midsurface to the reference surface containing the element's nodes. Since no element level calculations are performed for rigid elements, a specified offset affects only the handling of contact pairs with rigid surfaces formed by rigid elements.

This parameter accepts positive or negative values or the labels SPOS or SNEG. The positive values of the offset are in the direction of the element normal. When OFFSET=0.5 (or SPOS), the top surface of the rigid element is the reference surface. When OFFSET=−0.5 (or SNEG),

the bottom surface of the rigid element is the reference surface. The default is OFFSET=0, which indicates that the middle surface of the rigid element is the reference surface.

There are no data lines associated with this option in an Abaqus/Standard analysis.

Data line for R2D2 elements in an Abaqus/Explicit analysis:

First (and only) line:

1. Cross-sectional area of the element. The default is 0.

Data line for RAX2, R3D3, and R3D4 elements in an Abaqus/Explicit analysis:

First (and only) line:

1. Thickness of the element. The default is 0.

17.19 *RIGID SURFACE: Define an analytical rigid surface.

This option must be used when defining the seabed for three-dimensional drag chain elements in Abaqus/Standard analyses. For all other cases the preferred options for defining analytical rigid surfaces are the *SURFACE and the *RIGID BODY options.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Part module

References:

- "Surfaces: overview," Section 2.3.1 of the Abaqus Analysis User's Manual
- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Manual
- "Drag chains," Section 29.12.1 of the Abaqus Analysis User's Manual
- "RSURFU," Section 1.1.16 of the Abaqus User Subroutines Reference Manual
- *SURFACE

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the IRS-type elements or the three-dimensional drag chain elements that may interact with the rigid surface being defined.

The ELSET and NAME parameters are mutually exclusive.

NAME

Set this parameter equal to a label that will be used to refer to the rigid surface being created. This surface name is used to define contact interaction with another surface through the *CONTACT PAIR option.

The ELSET and NAME parameters are mutually exclusive.

REF NODE

Set this parameter equal to either the node number of the rigid body reference node or the name of a node set containing the rigid body reference node. If the name of a node set is chosen, the node set must contain exactly one node.

This parameter is relevant only when the NAME parameter is used.

TYPE

Set TYPE=SEGMENTS to create a two-dimensional rigid surface in the (X, Y) plane for planar models or in the (r, z) plane for axisymmetric models by defining connected line segments.

Set TYPE=CYLINDER to define a three-dimensional rigid surface by providing connected line segments and then sweeping them along a specified generator vector.

Set TYPE=REVOLUTION to define a three-dimensional rigid surface by providing connected line segments, which are given in an (r, z) plane and are rotated about an axis.

Set TYPE=USER to define a rigid surface via user subroutine **RSURFU**.

Optional parameter:

FILLET RADIUS

This parameter can be used with TYPE=SEGMENTS, TYPE=CYLINDER, or TYPE=REVOLUTION to define a radius of curvature to smooth discontinuities between adjoining straight-line segments, adjoining circular-arc segments, and adjoining straight-line and circular-arc segments. It has no effect on rigid surfaces defined with TYPE=USER.

No data lines are needed for TYPE=USER.

Data lines to define surfaces created with TYPE=SEGMENTS:

First line:

1. The "word" START.
2. Global *X*-coordinate or *r*-coordinate of the starting point of the line segments.
3. Global *Y*-coordinate or *z*-coordinate of the starting point of the line segments.

Second and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the rigid surface.

Data lines to define surfaces created with TYPE=CYLINDER:

First line:

1. Global *X*-coordinate of point *a*, the origin of the local (x, y) system (see Figure 17.19–1).
2. Global *Y*-coordinate of point *a*, the origin of the local (x, y) system.
3. Global *Z*-coordinate of point *a*, the origin of the local (x, y) system.
4. Global *X*-coordinate of point *b* on the local *x*-axis.
5. Global *Y*-coordinate of point *b* on the local *x*-axis.
6. Global *Z*-coordinate of point *b* on the local *x*-axis.

Second line:

1. Global *X*-coordinate of point *c* on the local cylinder generator vector.

2. Global *Y*-coordinate of point *c* on the local cylinder generator vector.
3. Global *Z*-coordinate of point *c* on the local cylinder generator vector.

Third line:

1. The "word" START.
2. Local *x*-coordinate of the starting point of the line segments.
3. Local *y*-coordinate of the starting point of the line segments.

Fourth and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the rigid surface.

Data lines to define surfaces created with TYPE=REVOLUTION:

First line:

1. Global *X*-coordinate of point *a*, the origin of the local (r, z) system (see Figure 17.19–2).
2. Global *Y*-coordinate of point *a*, the origin of the local (r, z) system.
3. Global *Z*-coordinate of point *a*, the origin of the local (r, z) system.
4. Global *X*-coordinate of point *b* on the symmetry axis (the local *z*-axis).
5. Global *Y*-coordinate of point *b* on the symmetry axis (the local *z*-axis).
6. Global *Z*-coordinate of point *b* on the symmetry axis (the local *z*-axis).

Second line:

1. The "word" START.
2. Local *r*-coordinate of the starting point of the line segments.
3. Local *z*-coordinate of the starting point of the line segments.

Third and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the rigid surface.

Data lines that define the line segments that form the rigid surface for TYPE=SEGMENTS, TYPE=CYLINDER, and TYPE=REVOLUTION:

Data line to define a straight line segment:

1. The "word" LINE.
2. *x*-coordinate of the endpoint of the line.
3. *y*-coordinate of the endpoint of the line.

Data line to define a circular arc segment (the arc must be less than 180°):

1. The "word" CIRCL.
2. *x*-coordinate of the endpoint of the circular arc.

3. y-coordinate of the endpoint of the circular arc.
4. x-coordinate of the center (origin) of the circular arc.
5. y-coordinate of the center (origin) of the circular arc.

Data line to define a parabolic arc segment:

1. The "word" PARAB.
2. x-coordinate of the middle point along the parabolic arc.
3. y-coordinate of the middle point along the parabolic arc.
4. x-coordinate of the endpoint of the parabolic arc.
5. y-coordinate of the endpoint of the parabolic arc.

For rigid surfaces created with TYPE=SEGMENTS, the x- and y-coordinates are the global X- and Y-coordinates or r- and z-coordinates. For rigid surfaces created with TYPE=CYLINDER, the x- and y-coordinates are the local x- and y-coordinates. For rigid surfaces created with TYPE=REVOLUTION, the x- and y-coordinates are the local r- and z-coordinates.

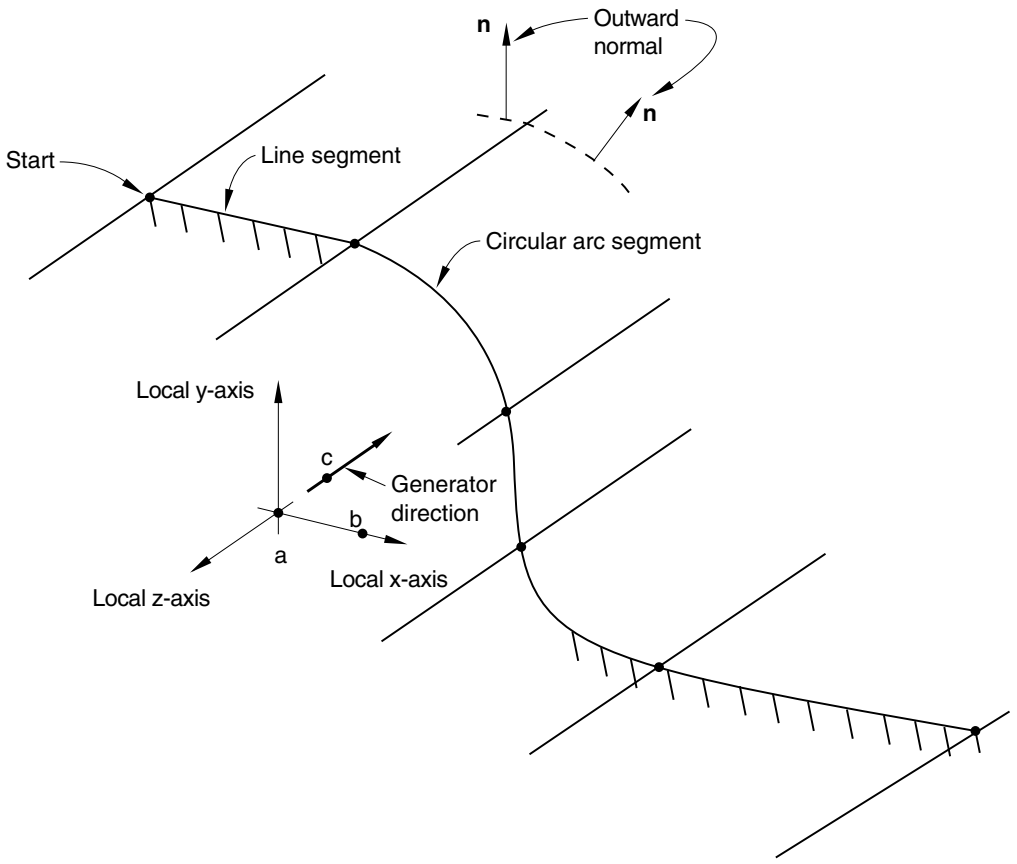


Figure 17.19–1 *RIGID SURFACE, TYPE=CYLINDER.

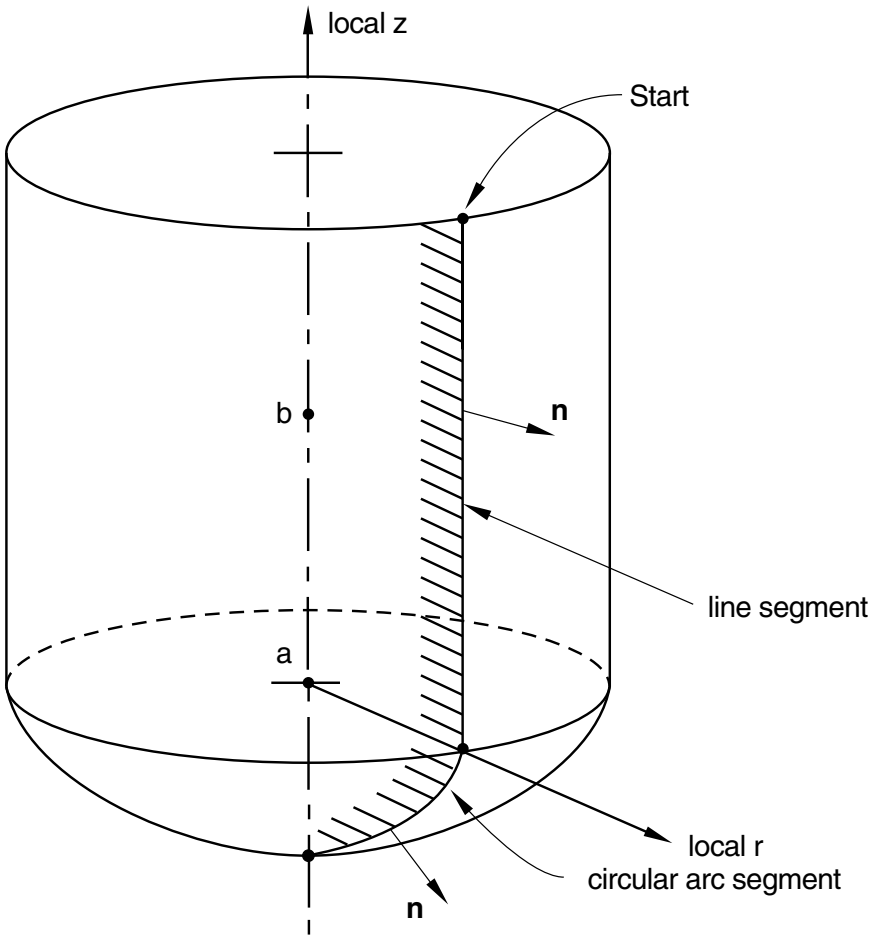


Figure 17.19–2 *RIGID SURFACE, TYPE=REVOLUTION.

17.20 *ROTARY INERTIA: Define rigid body rotary inertia.

This option is used to define rigid body rotary inertia values associated with ROTARYI elements.

It is also used in Abaqus/Standard analyses to define mass proportional damping (for direct-integration dynamic analysis) and composite damping (for modal dynamic analysis) associated with ROTARYI elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Property module and Interaction module.

Reference:

- "Rotary inertia," Section 27.2.1 of the Abaqus Analysis User's Manual

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the ROTARYI elements for which the value is being given.

Optional parameters:

ALPHA

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the α_R factor to create inertia proportional damping for the ROTARYI elements when used in direct-integration dynamics. This value is ignored in modal dynamics. The default is 0.0.

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the fraction of critical damping to be used with the ROTARYI elements when calculating composite damping factors for the modes when used in modal dynamics. This value is ignored in direct-integration dynamics. The default is 0.0.

ORIENTATION

Set this parameter equal to the name of an *ORIENTATION option ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) that is being used to define the directions of the local axes for which the rotary inertia values are being given. If the ORIENTATION parameter is omitted, it

is assumed that the components of the inertia tensor are being given with respect to the global axes; i.e., the global and local inertia axes coincide.

In large-displacement analysis (an Abaqus/Explicit analysis or when the NLGEOM parameter is included on the *STEP option in an Abaqus/Standard analysis), the local axes of inertia rotate with the rotation of the node to which the ROTARYI element is attached.

Data line to define the rotary inertia:

First (and only) line:

1. Rotary inertia about the local 1-axis, I_{11}.
2. Rotary inertia about the local 2-axis, I_{22}.
3. Rotary inertia about the local 3-axis, I_{33}.
4. Product of inertia, I_{12}.
5. Product of inertia, I_{13}.
6. Product of inertia, I_{23}.

The rotary inertia should be given in units of ML^2. Abaqus does not use any specific physical units, so the user's choice must be consistent.

18. S

18.1 *SECTION CONTROLS: Specify section controls.

WARNING: Using values larger than the default values for hourglass control can produce excessively stiff response and sometimes can even lead to instability if the values are too large. Hourglassing that occurs with the default hourglass control parameters is usually an indication that the mesh is too coarse. Therefore, it is generally better to refine the mesh than to add stronger hourglass control.

This option is used to choose a nondefault hourglass control approach for reduced-integration elements in Abaqus/Standard and Abaqus/Explicit and modified tetrahedral or triangular elements in Abaqus/Standard, to scale the default coefficients used in the hourglass control, or to activate distortion control for solid elements. In Abaqus/Explicit it is also used to select a nondefault kinematic formulation for 8-node brick elements, to choose the second-order accurate formulation for solids and shells, to turn off the drill stiffness in small-strain shell elements S3RS and S4RS, to introduce the stresses in membrane elements from the initial configuration gradually into the model, to scale the linear and quadratic bulk viscosities, to specify whether elements must be deleted when they are completely damaged, or to specify a value of the scalar degradation parameter at or above which elements are assumed to be completely damaged. It is used in conjunction with the *MEMBRANE SECTION, *SHELL GENERAL SECTION, *SHELL SECTION, and/or *SOLID SECTION options. It can be used with *BEAM SECTION and *BEAM GENERAL SECTION to scale the linear and bulk viscosities. It can also be used with the *SOLID SECTION, *SHELL SECTION, *MEMBRANE SECTION, *COHESIVE SECTION, and *CONNECTOR SECTION options to specify whether elements must be deleted when they are completely damaged, to specify a value of the scalar degradation parameter at or above which elements are assumed to be completely damaged, or, in Abaqus/Standard, to specify the viscosity coefficient that controls the viscous regularization.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Mesh module

References:

- "Section controls," Section 24.1.4 of the Abaqus Analysis User's Manual
- *COHESIVE SECTION
- *CONNECTOR SECTION
- *MEMBRANE SECTION
- *SHELL GENERAL SECTION
- *SHELL SECTION
- *SOLID SECTION

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the section control definition. All section control names in the same input file must be unique.

Optional parameters:

DISTORTION CONTROL

This parameter applies to Abaqus/Explicit analyses and to solid sections in Abaqus/Standard analyses containing C3D10I elements.

Set DISTORTION CONTROL=YES to activate a constraint that acts to prevent negative element volumes or other excessive distortion for crushable materials. This is the default value for elements with hyperelastic or hyperfoam materials. The DISTORTION CONTROL parameter is not relevant for linear kinematics and cannot prevent elements from being distorted due to temporal instabilities, hourglass instabilities, or physically unrealistic deformation.

Set DISTORTION CONTROL=NO to deactivate a constraint that acts to prevent negative element volumes or other excessive distortion for crushable materials. This value is the default except for elements with hyperelastic or hyperfoam materials.

DRILL STIFFNESS

This parameter applies only to the small-strain shell elements S3RS and S4RS.

Set DRILL STIFFNESS=ON (default) to activate the constraint against the drill mode in S3RS and S4RS elements.

Set DRILL STIFFNESS=OFF to deactivate the constraint against the drill mode. Deactivating the drill constraint can result in large values for the rotation degrees of freedom at the nodes of these elements. Use this setting if you need to match the element formulation offered in Abaqus/Explicit 6.9 and earlier for these elements.

The drill constraint is always active for finite-strain conventional shell elements such as S4R.

ELEMENT DELETION

This parameter applies to all elements with progressive damage behavior, except connector elements in Abaqus/Explicit.

Set ELEMENT DELETION=YES to allow element deletion when the element has completely damaged. This value is the default for all elements with a damage evolution model. However, this value is not applicable to pore pressure cohesive elements.

Set ELEMENT DELETION=NO to allow fully damaged elements to remain in the computations. The element retains a residual stiffness given by the specified value of MAX DEGRADATION. In addition, elements with three-dimensional stress states (including generalized plane strain elements) can sustain hydrostatic compressive stresses, and elements with one-dimensional stress states can sustain compressive stresses. This value is the default for pore pressure cohesive elements and is not available for beam elements.

HOURGLASS

Set HOURGLASS=COMBINED to define the viscous-stiffness form of hourglass control for solid and membrane elements with reduced integration in Abaqus/Explicit.

Set HOURGLASS=ENHANCED (default for elements with hyperelastic and hyperfoam materials in Abaqus/Standard and Abaqus/Explicit; default in Abaqus/Standard and only option in Abaqus/Explicit for modified tetrahedral or triangular elements) to define hourglass control that is based on the assumed enhanced strain method for solid, membrane, finite-strain shell elements with reduced integration and modified tetrahedral or triangular elements in Abaqus/Standard and Abaqus/Explicit. Any data given on the data line will be ignored for this case.

Set HOURGLASS=RELAX STIFFNESS (default for Abaqus/Explicit, except for elements with hyperelastic and hyperfoam materials) to use the integral viscoelastic form of hourglass control for all elements with reduced integration in Abaqus/Explicit.

Set HOURGLASS=STIFFNESS (default for Abaqus/Standard, except for elements with hyperelastic and hyperfoam materials and modified tetrahedral or triangular elements) to define hourglass control that is strictly elastic for all elements with reduced integration in Abaqus/Standard and Abaqus/Explicit and modified tetrahedral or triangular elements in Abaqus/Standard.

Set HOURGLASS=VISCOUS to define the hourglass damping used to control the hourglass modes for solid and membrane elements with reduced integration in Abaqus/Explicit.

INITIAL GAP OPENING

This parameter applies only to Abaqus/Standard analyses using pore pressure cohesive elements.

Set this parameter equal to the value of the initial gap opening used in the tangential flow continuity equation for pore pressure cohesive elements. The default value is 0.002.

KINEMATIC SPLIT

Include this parameter to change the kinematic formulation for 8-node brick elements only.

Set KINEMATIC SPLIT=AVERAGE STRAIN (default in Abaqus/Explicit) to use the uniform strain formulation and the hourglass shape vectors in the hourglass control. This is the only option available for Abaqus/Standard.

Set KINEMATIC SPLIT=CENTROID to use the centroid strain formulation and the hourglass base vectors in the hourglass control in Abaqus/Explicit.

Set KINEMATIC SPLIT=ORTHOGONAL to use the centroid strain formulation and the hourglass shape vectors in the hourglass control in Abaqus/Explicit.

If SECOND ORDER ACCURACY=YES, the KINEMATIC SPLIT parameter will be reset to AVERAGE STRAIN in Abaqus/Explicit.

LENGTH RATIO

This parameter applies only to Abaqus/Explicit analyses and is valid only when the DISTORTION CONTROL parameter is used.

Set this parameter equal to r ($0.0 < r \leq 1.0$) to define the length ratio when using distortion control for crushable materials. The default value is $r = 0.1$.

MAX DEGRADATION

This parameter applies to all elements with progressive damage behavior, except connector elements in Abaqus/Explicit.

Set this parameter equal to the value of the damage variable at or above which a material point will be assumed to be completely damaged. This parameter also determines the amount of residual stiffness that will be retained by elements for which the ELEMENT DELETION parameter is set to NO. For elements other than cohesive elements, connector elements, and elements with plane stress formulations the default value is 1.0 if the element is deleted from the mesh and 0.99 otherwise. For cohesive elements, connector elements, and elements with plane stress formulations the default value is always 1.0.

PERTURBATION

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to a small perturbation to be applied to the second orientation for the FLEXION-TORSION connectors.

RAMP INITIAL STRESS

This parameter applies to membrane elements in Abaqus/Explicit analyses.

Set this parameter equal to the name of a total time-based amplitude defined to go from an initial value of zero to a final value of one. When this parameter is specified, the element stiffness is controlled until the amplitude value reaches its final value of one, so that the initial stresses are introduced gradually and not abruptly.

SECOND ORDER ACCURACY

Set SECOND ORDER ACCURACY=YES to use a second-order accurate formulation for solid or shell elements suitable for problems undergoing a large number of revolutions (> 5) in Abaqus/Explicit. This is the only option available for Abaqus/Standard.

Set SECOND ORDER ACCURACY=NO (default in Abaqus/Explicit) to use the first-order accurate solid or shell elements in Abaqus/Explicit.

The SECOND ORDER ACCURACY parameter is not relevant for linear kinematics.

VISCOSITY

This parameter applies to cohesive elements, connector elements, and elements with plane stress formulations (plane stress, shell, continuum shell, and membrane elements) in Abaqus/Standard analyses.

Set this parameter equal to the value of the viscosity coefficient used in the viscous regularization scheme for cohesive elements or connector elements or equal to the value of the damping coefficient used in connector failure modeling. When this parameter is used to specify the viscosity coefficients for the damage model for fiber-reinforced materials, the specified value is applied to all the damage modes. The default value is 0.0.

WEIGHT FACTOR

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to α ($0.0 \leq \alpha \leq 1.0$) to scale the contributions from the constant hourglass stiffness term and the hourglass damping term to the hourglass control formulation. Setting $\alpha = 0.0$ or $\alpha = 1.0$ corresponds to pure constant stiffness hourglass control and pure damping hourglass control, respectively. The default is $\alpha = 0.5$. This option is used only for solid and membrane elements when the HOURGLASS parameter is set equal to COMBINED.

Data line to define the hourglass control and bulk viscosity:

First (and only) line:

1. Scaling factor, s^s, for the hourglass stiffness, for use with displacement degrees of freedom. If this value is left blank, Abaqus will use the default value of 1.0. The suggested range for the value of s^s is between 0.2 and 3.0.
2. Scaling factor, s^r, for the hourglass stiffness, for use with rotational degrees of freedom in shell elements. If this value is left blank, Abaqus will use the default value of 1.0. The suggested range for the value of s^r is between 0.2 and 3.0.
3. Scaling factor, s^w, for the hourglass stiffness for use with the out-of-plane displacement degree of freedom in small-strain shell elements in Abaqus/Explicit. If this value is left blank, Abaqus/Explicit will use the default value of 1.0. The suggested range for the value of s^w is between 0.2 and 3.0. This scaling factor is not relevant for Abaqus/Standard.
4. Scaling factor for the linear bulk viscosity in Abaqus/Explicit. If this value is left blank, Abaqus/Explicit will use the default value of 1.0. The suggested range for the value is between 0.0 and 1.0. This scaling factor is not relevant for Abaqus/Standard.
5. Scaling factor for the quadratic bulk viscosity in Abaqus/Explicit. If this value is left blank, Abaqus/Explicit will use the default value of 1.0. The suggested range for the value is between 0.0 and 1.0. This scaling factor is not relevant for Abaqus/Standard.
6. Scaling factor, s^p, for the hourglass stiffness for use with pressure Lagrange multiplier degrees of freedom for element type C3D4H in Abaqus/Standard. If this value is left blank, Abaqus/Standard will use the default value of 1.0. The suggested range for the value of s^p is between 0.2 and 3.0. This scaling factor is not relevant for Abaqus/Explicit.

18.2 *SECTION FILE: Define results file requests of accumulated quantities on user-defined surface sections.

This option is used to control output to the results file of accumulated quantities associated with a user-defined section. Depending on the analysis type the output may include one or several of the following: the total force, the total moment, the total heat flux, the total current, the total mass flow, or the total pore fluid volume flux associated with the section. This option is not available for eigenfrequency extraction, eigenvalue buckling prediction, complex eigenfrequency extraction, or linear dynamics procedures.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Manual
- "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual

Required parameters:

NAME

Set this parameter equal to a label that will be used to identify the output for the section. Section names in the same input file must be unique.

SURFACE

Set this parameter equal to the name used in the *SURFACE option to define the surface.

Optional parameters:

AXES

Set AXES=LOCAL if output is desired in the local coordinate system. Set AXES=GLOBAL (default) to output quantities in the global coordinate system.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

UPDATE

Set UPDATE=NO if output is desired in the original local system of coordinates. Set UPDATE=YES (default) to output quantities in a local system of coordinates that rotates with the

average rigid body motion of the surface section. This parameter is relevant only if AXES=LOCAL and the NLGEOM parameter is active in the step.

Optional data lines:

First line:

1. Node number of the anchor point (blank if coordinates given).
2. First coordinate of the anchor point (ignored if node number given).
3. Second coordinate of the anchor point (ignored if node number given).
4. Third coordinate of the anchor point (for three-dimensional cases only; ignored if node number given).

Leave this line blank to allow Abaqus to define the anchor point.

Second line:

1. Node number used to specify point *a* in Figure 18.2–1 (blank if coordinates given).
2. First coordinate of point *a* (ignored if node number given).
3. Second coordinate of point *a* (ignored if node number given).

The remaining data items are relevant only for three-dimensional cases.

4. Third coordinate of point *a* (ignored if node number given).
5. Node number used to specify point *b* (blank if coordinates given).
6. First coordinate of point *b* (ignored if node number given).
7. Second coordinate of point *b* (ignored if node number given).
8. Third coordinate of point *b* (ignored if node number given).

Leave this line blank to allow Abaqus to define the axes.

Third line:

1. Give the identifying keys for the variables to be output. The keys are defined in the "Section variables" section of "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual.

Omit both the first and second data lines for AXES=GLOBAL or to allow Abaqus to define the anchor point and the axes for AXES=LOCAL. Repeat the third data line as often as necessary to define the variables to be written to the results file. If this line is omitted, all appropriate variables ("Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Manual) will be output.

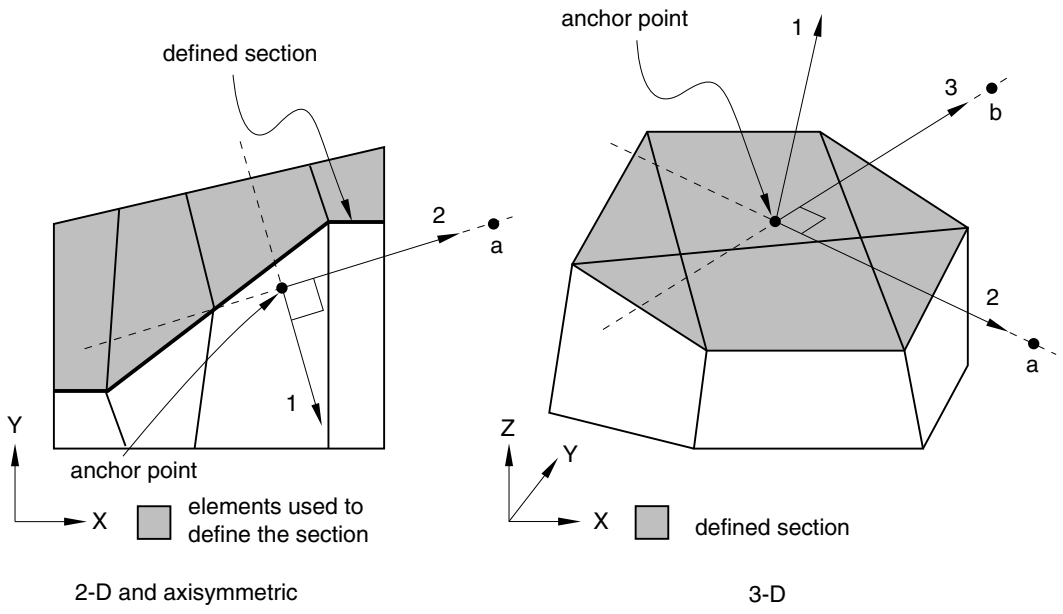


Figure 18.2–1 User-defined local coordinate system.

18.3 *SECTION ORIGIN: Define a meshed cross-section origin.

This option is used in conjunction with the *BEAM SECTION GENERATE option to define the location of the beam node on a meshed beam cross-section.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Meshed beam cross-sections," Section 10.5.1 of the Abaqus Analysis User's Manual
- *BEAM SECTION GENERATE

Optional parameter:

ORIGIN

Set ORIGIN=CENTROID to place the beam node at the section centroid.
Set ORIGIN=SHEAR CENTER to place the beam node at the shear center.

Data line to define the location of the beam node if the ORIGIN parameter is omitted:

First (and only) data line:

1. Global x-coordinate of the beam node. The default is 0.0.
2. Global y-coordinate of the beam node. The default is 0.0.

If this option is not used, the section origin is placed at the origin of the coordinate system on the meshed cross-section model. If this option is used without the ORIGIN parameter and no data line is given, the section origin is placed at the centroid.

18.4 *SECTION POINTS: Locate points in the beam section for which stress and strain output are required.

This option is used as model data in Abaqus/Standard and Abaqus/Explicit in conjunction with the *BEAM GENERAL SECTION option and as history data in Abaqus/Standard in conjunction with the *BEAM SECTION GENERATE option.

When used in conjunction with the *BEAM GENERAL SECTION option and a predefined library section, it locates section points in the beam section for which axial stress and axial strain output are required.

When used in conjunction with the *BEAM GENERAL SECTION, SECTION=MESHED option, it locates elements and integration points in the beam cross-section model for which stress and strain output are required and provides material data and derivatives of the warping function required for determining stress and strain output.

When used in conjunction with the *BEAM SECTION GENERATE option, it associates a user-defined section point label with elements and integration points in the beam cross-section model for which stress and strain output are required during the subsequent beam analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Part, Part instance, Step

Abaqus/CAE: Property module

References:

- "Using a general beam section to define the section behavior," Section 26.3.7 of the Abaqus Analysis User's Manual
- "Meshed beam cross-sections," Section 10.5.1 of the Abaqus Analysis User's Manual
- *BEAM GENERAL SECTION

There are no parameters associated with this option.

Data lines to locate section points for output when used in conjunction with the *BEAM GENERAL SECTION option and a predefined library section:

First line:

1. Local x_1-position of first section point.
2. Local x_2-position of first section point.

3. Local x_1-position of second section point.
4. Local x_2-position of second section point.

Continue giving (x_1, x_2) coordinate pairs for as many points as needed. At most four pairs of points can be specified on any data line. If the point (0,0) is specified as the last entry on a line, it will be ignored unless it is the only point requested.

Data lines to locate elements and integration point numbers for meshed sections when used in conjunction with the *BEAM GENERAL SECTION, SECTION=MESHED option:

First line:

1. Section point label.
2. Two-dimensional model element number.
3. Integration point number.

Second line:

1. Young's modulus, E.
2. Shear modulus in user-defined 1-direction, G_1.
3. Shear modulus in user-defined 2-direction, G_2 (for isotropic materials G_2=G_1).
4. Material orientation angle, α, measured counterclockwise from the beam local 1-direction.
5. Local $-(x_1 - x_{1c})$-position of the integration point with respect to the centroid.
6. Local $(x_2 - x_{2c})$-position of the integration point with respect to the centroid.
7. Derivative of the warping function $(\frac{\partial \Psi}{\partial x_1} - (x_2 - x_{2s}))$ with respect to the local 1-direction and the shear center.
8. Derivative of the warping function $(\frac{\partial \Psi}{\partial x_2} + (x_1 - x_{1s}))$ with respect to the local 2-direction and the shear center.

Repeat this set of data lines for as many integration points as needed.

Data lines to locate elements and integration point numbers in the cross-section model when used in conjunction with the *BEAM SECTION GENERATE option:

First line:

1. Section point label.
2. Element number.
3. Integration point number.

Repeat this data line as often as necessary to specify the elements and integration points.

18.5 *SECTION PRINT: Define print requests of accumulated quantities on user-defined surface sections.

This option is used to provide tabular output of accumulated quantities associated with a user-defined section. Depending on the analysis type the output may include one or several of the following: the total force, the total moment, the total heat flux, the total current, the total mass flow, or the total pore fluid volume flux associated with the section. This option is not available for eigenfrequency extraction, eigenvalue buckling prediction, complex eigenfrequency extraction, or linear dynamics procedures.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Manual
- "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual

Required parameters:

NAME

Set this parameter equal to a label that will be used to identify the output for the section. Section names in the same input file must be unique.

SURFACE

Set this parameter equal to the name used in the *SURFACE option to define the surface.

Optional parameters:

AXES

Set AXES=LOCAL if output is desired in the local coordinate system. Set AXES=GLOBAL (default) to output quantities in the global coordinate system.

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

UPDATE

Set UPDATE=NO if output is desired in the original local system of coordinates. Set UPDATE=YES (default) to output quantities in a local system of coordinates that rotates with the

average rigid body motion of the surface section. This parameter is relevant only if AXES=LOCAL and the NLGEOM parameter is active in the step.

Optional data lines:

First line:

1. Node number of the anchor point (blank if coordinates given).
2. First coordinate of the anchor point (ignored if node number given).
3. Second coordinate of the anchor point (ignored if node number given).
4. Third coordinate of the anchor point (for three-dimensional cases only; ignored if node number given).

Leave this line blank to allow Abaqus to define the anchor point.

Second line:

1. Node number used to specify point *a* in Figure 18.5–1 (blank if coordinates given).
2. First coordinate of point *a* (ignored if node number given).
3. Second coordinate of point *a* (ignored if node number given).

The remaining data items are relevant only for three-dimensional cases.

4. Third coordinate of point *a* (ignored if node number given).
5. Node number used to specify point *b* (blank if coordinates given)
6. First coordinate of point *b* (ignored if node number given).
7. Second coordinate of point *b* (ignored if node number given).
8. Third coordinate of point *b* (ignored if node number given).

Leave this line blank to allow Abaqus to define the axes.

Third line:

1. Give the identifying keys for the variables to be output. The keys are defined in the "Section variables" section of "Abaqus/Standard output variable identifiers," Section 4.2.1 of the Abaqus Analysis User's Manual.

Omit both the first and second data lines for AXES=GLOBAL or to allow Abaqus to define the anchor point and the axes for AXES=LOCAL. Repeat the third data line as often as necessary: each line defines a table. If this line is omitted, all appropriate variables ("Output to the data and results files," Section 4.1.2 of the Abaqus Analysis User's Manual) will be output.

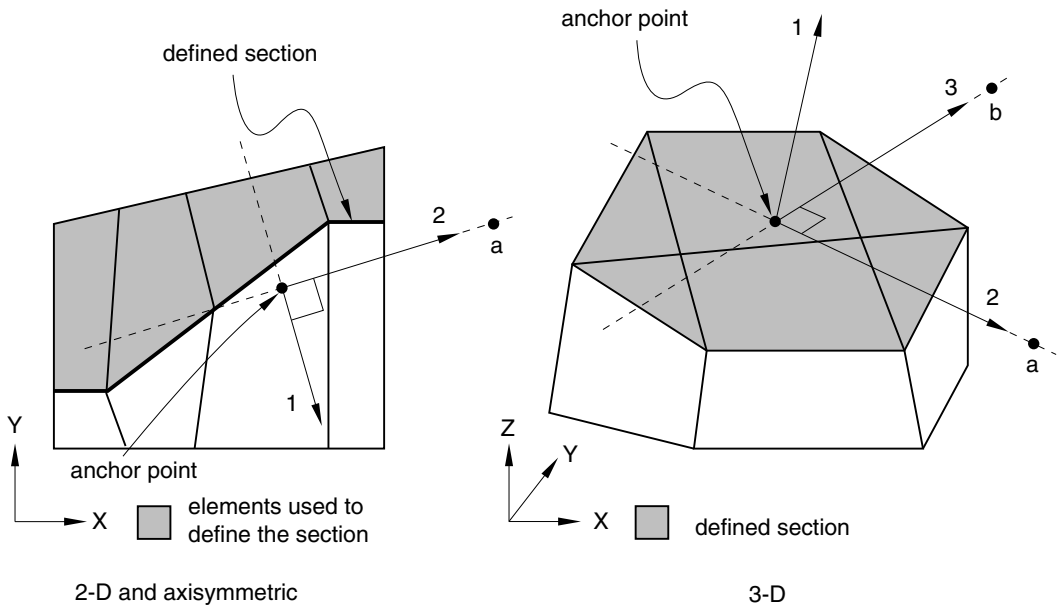


Figure 18.5–1 User-defined local coordinate system.

18.6 *SELECT CYCLIC SYMMETRY MODES: Specify the cyclic symmetry modes in an eigenvalue analysis of a cyclic symmetric structure.

This option is used to specify which cyclic symmetry modes should be used in an eigenvalue analysis.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Manual
- *CYCLIC SYMMETRY MODEL
- *TIE

Optional parameters:

EVEN

Include this parameter to request the even cyclic symmetry modes only. If this parameter is omitted, all cyclic symmetry modes will be used within the NMIN, NMAX range.

NMIN

Set this parameter equal to the lowest cyclic symmetry mode number. The default value is 0.

NMAX

Set this parameter equal to the highest cyclic symmetry mode number. The default value is the highest number that is feasible for the number of sectors given in the *CYCLIC SYMMETRY MODEL option.

There are no data lines associated with this option.

18.7 *SELECT EIGENMODES: Select the modes to be used in a modal dynamic analysis.

This option selects the modes to be used in a dynamic analysis based on modes or in a complex eigenvalue extraction analysis. Only one option per step can be used. If this option is not used, all modes extracted in the prior *FREQUENCY step will be used, including residual modes if they were activated.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Implicit dynamic analysis using direct integration," Section 6.3.2 of the Abaqus Analysis User's Manual
- "Complex eigenvalue extraction," Section 6.3.6 of the Abaqus Analysis User's Manual
- "Transient modal dynamic analysis," Section 6.3.7 of the Abaqus Analysis User's Manual
- "Mode-based steady-state dynamic analysis," Section 6.3.8 of the Abaqus Analysis User's Manual
- "Subspace-based steady-state dynamic analysis," Section 6.3.9 of the Abaqus Analysis User's Manual
- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Manual
- "Response spectrum analysis," Section 6.3.10 of the Abaqus Analysis User's Manual
- "Random response analysis," Section 6.3.11 of the Abaqus Analysis User's Manual
- *MODAL DAMPING

Optional parameters:

DEFINITION

Set DEFINITION=MODE NUMBERS (default) to indicate that the selected modes are given as a collection of mode numbers.

Set DEFINITION= FREQUENCY RANGE to indicate that the modes are selected from the specified frequency ranges including frequency boundaries. Frequency ranges can be discontinuous.

If both the *MODAL DAMPING and *SELECT EIGENMODES options are used in the same step, the DEFINITION parameter must be set equal to the same value in both options.

GENERATE

If this parameter is included, each data line should give a first mode, n_1; a last mode, n_2; and the increment in mode numbers between these modes, i. Then all modes going from n_1 to n_2 in steps of i will be added to the set. i must be an integer such that $(n_2 - n_1)/i$ is a whole number (not a fraction).

This parameter can be used only with DEFINITION=MODE NUMBERS.

Data lines if DEFINITION=MODE NUMBERS and the GENERATE parameter is omitted:

First line:

1. List of modes to be used.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines if DEFINITION=MODE NUMBERS and the GENERATE parameter is included:

First line:

1. First mode number.
2. Last mode number.
3. Increment in mode numbers between modes. The default is 1.

Repeat this data line as often as necessary.

Data lines if DEFINITION=FREQUENCY RANGE:

First line:

1. Lower boundary of the frequency range (in cycles/time).
2. Upper boundary of the frequency range (in cycles/time).

Repeat this data line as often as necessary.

18.8 *SFILM: Define film coefficients and associated sink temperatures over a surface for heat transfer analysis.

This option is used to provide film coefficients and sink temperatures over a surface for fully coupled thermal-stress analysis. In Abaqus/Standard it is also used in heat transfer and coupled thermal-electrical analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual
- "FILM," Section 1.1.6 of the Abaqus User Subroutines Reference Manual

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the sink temperature, θ^0, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference sink temperature is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option ("Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted in an Abaqus/Explicit analysis, the reference sink temperature is applied immediately at the beginning of the step.

For nonuniform films of type FNU (which are available only in Abaqus/Standard), the sink temperature amplitude is defined in user subroutine **FILM**, and AMPLITUDE references are ignored.

FILM AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the film coefficient, h, with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference film coefficient is applied immediately at the beginning of the step and kept constant over the step, independent of the value assigned to the AMPLITUDE parameter on the *STEP option. If this parameter is omitted in an Abaqus/Explicit analysis, the reference sink temperature given on the data lines is applied throughout the step.

The FILM AMPLITUDE parameter is ignored if a nonuniform film coefficient is defined in user subroutine **FILM** or if a film coefficient is defined to be a function of temperature and field variables using the *FILM PROPERTY option.

OP

Set OP=MOD (default) to modify existing films or to define additional films.
Set OP=NEW if all existing *SFILMs applied to the model should be removed.

Data lines to define sink temperatures and film coefficients:

First line:

1. Surface name.
2. Film type label F or FNU.
3. Reference sink temperature value, θ^0. (Units of θ.) For nonuniform film coefficients the sink temperature must be defined in user subroutine **FILM**. If given, this value will be passed into the user subroutine.
4. Reference film coefficient value, h (units of $JT^{-1}L^{-2}\theta^{-1}$), or name of the film property table defined with the *FILM PROPERTY option. Nonuniform film coefficients must be defined in user subroutine **FILM**. If given, this value will be passed into the user subroutine.

Repeat this data line as often as necessary to define film conditions for different surfaces.

18.9 *SFLOW: Define seepage coefficients and associated sink pore pressures normal to a surface.

This option is used to provide seepage coefficients and sink pore pressures to control pore fluid flow normal to the surface in consolidation analysis.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Pore fluid flow," Section 30.4.6 of the Abaqus Analysis User's Manual
- "FLOW," Section 1.1.7 of the Abaqus User Subroutines Reference Manual

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that gives the variation of reference pore pressure with time. If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step depending on the value assigned to the AMPLITUDE parameter on the *STEP option (see "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). The AMPLITUDE parameter is ignored for nonuniform seepage flow boundary conditions defined in user subroutine **FLOW** and for drainage-only seepage boundary conditions.

OP

Set OP=MOD (default) to modify existing flows or to define additional flows.

Set OP=NEW if all existing *SFLOWs applied to the model should be removed. New flows can be defined.

Data lines to define uniform seepage:

First line:

1. Surface name.
2. Seepage flow type label Q.
3. Reference pore pressure value, u_w^{∞}. (Units of FL^{-2}.)
4. Reference seepage coefficient value, k_s. (Units of $\mathrm{F}^{-1}\mathrm{L}^3\mathrm{T}^{-1}$.)

Repeat this data line as often as necessary to define uniform seepage for various surfaces.

Data lines to define drainage-only seepage:

First line:

1. Surface name.
2. Seepage flow type label QD.
3. Drainage-only seepage coefficient value, k_s. (Units of $\mathrm{F^{-1}L^3T^{-1}}$.)

Repeat this data line as often as necessary to define drainage-only seepage for various surfaces.

Data lines to define nonuniform seepage:

First line:

1. Surface name.
2. Seepage flow type label QNU.
3. Optional reference pore pressure value. If given, this value is passed into user subroutine **FLOW** in the variable used to define the sink pore pressure.
4. Optional reference seepage coefficient. If given, this value is passed into user subroutine **FLOW** in the variable used to define the seepage coefficient.

The reference pore pressure value, u_w^∞, and reference seepage coefficient, k_s, are defined in user subroutine **FLOW** for nonuniform flow.

Repeat this data line as often as necessary to define nonuniform seepage for various surfaces.

18.10 *SHEAR CENTER: Define the position of the shear center of a beam section.

This option can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=GENERAL or the *BEAM GENERAL SECTION, SECTION=MESHED option. It is used to define the position of the shear center of the section with respect to the local (1, 2) axis system.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- "Using a general beam section to define the section behavior," Section 26.3.7 of the Abaqus Analysis User's Manual
- *BEAM GENERAL SECTION
- "Meshed beam cross-sections," Section 10.5.1 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data line to define the shear center:

First (and only) line:

1. Local x_1-coordinate of shear center, x_{1s}. The default is 0.
2. Local x_2-coordinate of shear center, x_{2s}. The default is 0.

18.11 *SHEAR FAILURE: Specify a shear failure model and criterion.

This option is used with the Mises or the Johnson-Cook plasticity models to specify shear failure of the material. It must be used in conjunction with the option *PLASTIC, HARDENING=ISOTROPIC or JOHNSON COOK.

Product: Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Classical metal plasticity," Section 20.2.1 of the Abaqus Analysis User's Manual
- "Johnson-Cook plasticity," Section 20.2.7 of the Abaqus Analysis User's Manual
- "Dynamic failure models," Section 20.2.8 of the Abaqus Analysis User's Manual
- *PLASTIC

Optional parameters:

DEPENDENCIES

This parameter is relevant only for TYPE=TABULAR.

Set this parameter equal to the number of field variable dependencies included in the definition of the equivalent plastic strain at failure, in addition to temperature. If this parameter is omitted, it is assumed that the strain at failure depends on the plastic strain, the plastic strain rate, the dimensionless pressure-deviatoric stress ratio and, possibly, on temperature. See "Using the DEPENDENCIES parameter to define field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

ELEMENT DELETION

Set ELEMENT DELETION=YES (default) to allow element deletion when the failure criterion is met.

Set ELEMENT DELETION=NO to allow the element to take hydrostatic compressive stress only when the failure criterion is met.

TYPE

Set TYPE=JOHNSON COOK to define the Johnson-Cook shear failure criterion. It requires the use of the *PLASTIC, HARDENING=JOHNSON COOK option.

Set TYPE=TABULAR (default) to define the failure strain in tabular data form. It requires the use of the *PLASTIC, HARDENING=ISOTROPIC option.

Data lines to define the failure strain in tabular form (TYPE=TABULAR):

First line:

1. Equivalent plastic strain at failure, $\bar{\varepsilon}_f^{pl}$.
2. Rate of equivalent plastic strain, $\dot{\bar{\varepsilon}}^{pl}$.
3. Dimensionless pressure-deviatoric stress ratio, p/q.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of strain at failure on plastic strain, plastic strain rate, stress ratio and, if needed, on temperature and other predefined field variables.

Data line to define the Johnson-Cook shear failure criterion (TYPE=JOHNSON COOK):

First (and only) line:

1. Johnson-Cook failure parameter, d_1.
2. Johnson-Cook failure parameter, d_2.
3. Johnson-Cook failure parameter, d_3.
4. Johnson-Cook failure parameter, d_4.
5. Johnson-Cook failure parameter, d_5.

18.12 *SHEAR RETENTION: Define the reduction of the shear modulus associated with crack surfaces in a *CONCRETE model as a function of the tensile strain across the crack.

This option is used to give a multiplying factor, ϱ, that defines the modulus for shearing of cracks as a fraction of the elastic shear modulus of the uncracked concrete. If this option is used, it should follow the *CONCRETE option. The *SHEAR RETENTION option can also be used in conjunction with the *FAILURE RATIOS option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Concrete smeared cracking," Section 20.6.1 of the Abaqus Analysis User's Manual
- *CONCRETE
- *FAILURE RATIOS

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of shear retention behavior, in addition to temperature. If this parameter is omitted, it is assumed that the shear retention behavior depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define the shear retention behavior:

First line:

1. ϱ^{close}. The default is 1.0.
2. $\varepsilon^{\max}$. The default is a very large number (full shear retention).
3. Not used.
4. Not used.
5. Temperature.
6. First field variable.

7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the shear retention behavior on temperature and other predefined field variables.

18.13 *SHEAR TEST DATA: Used to provide shear test data.

This option can be used only in conjunction with the *VISCOELASTIC option. The *SHEAR TEST DATA option cannot be used for a viscoelastic material if the *COMBINED TEST DATA option is used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Using shear test data to define a viscoelastic material

References:

- "Time domain viscoelasticity," Section 19.7.1 of the Abaqus Analysis User's Manual
- *VISCOELASTIC

Optional parameter:

SHRINF

To specify creep test data, set this parameter equal to the value of the long-term, normalized shear compliance $j_S(\infty)$.

To specify relaxation test data, set this parameter equal to the value of the long-term, normalized shear modulus $g_R(\infty)$. The shear compliance and shear modulus are related by $j_S(\infty) = 1/g_R(\infty)$. The fitting procedure will use the specified value in the constraint $1 - \sum_{i=1}^{N} \bar{g}_i^P = g_R(\infty)$.

Data lines to specify creep test data:

First line:

1. Normalized shear compliance $j_S(t)$. $(j_S(t) \geq 1)$
2. Time t. $(t > 0)$

Repeat this data line as often as necessary to give the compliance-time data.

Data lines to specify relaxation test data:

First line:

1. Normalized shear relaxation modulus $g_R(t)$. $(0 \leq g_R(t) \leq 1)$

2. Time t. $(t > 0)$

Repeat this data line as often as necessary to give the modulus-time data.

18.14 *SHELL GENERAL SECTION: Define a general, arbitrary, elastic shell section.

This option is used to define a general, arbitrary, elastic shell section.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- "Shell elements: overview," Section 26.6.1 of the Abaqus Analysis User's Manual
- "Using a general shell section to define the section behavior," Section 26.6.6 of the Abaqus Analysis User's Manual
- "UGENS," Section 1.1.30 of the Abaqus User Subroutines Reference Manual

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the shell elements for which the section behavior is being defined.

Required parameter in Abaqus/Explicit, optional parameter in Abaqus/Standard:

DENSITY

Set this parameter equal to the mass per unit surface area of the shell.

If the MATERIAL and COMPOSITE parameters are omitted, this density accounts for the mass of the shell, since no material definition is given.

If the MATERIAL or COMPOSITE parameter is used, the mass of the shell includes a contribution from this parameter in addition to any contribution from the material definition.

Optional parameters:

BENDING ONLY

Include this parameter to ignore membrane stiffness effects in the shell. Bending stiffness coefficients are computed normally. Membrane-bending coupling coefficients are set to zero. Diagonal membrane stiffness coefficients are set to 1×10^{-6} times the largest diagonal bending stiffness term. Off-diagonal membrane stiffness coefficients are set to zero.

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see "Section controls," Section 24.1.4 of the Abaqus Analysis User's Manual) to be used to specify the second-order accurate element formulation option, a nondefault hourglass control formulation option, or scale factors.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition to be used to specify the enhanced hourglass control formulation (see "Section controls," Section 24.1.4 of the Abaqus Analysis User's Manual) or to be used in a subsequent Abaqus/Explicit import analysis.

LAYUP

This parameter is relevant only when the COMPOSITE parameter is used.

Set this parameter equal to the name of a composite layup (see Chapter 22, "Composite layups," of the Abaqus/CAE User's Manual). Abaqus/CAE uses this name to identify the composite layup that contains the shell section.

MEMBRANE ONLY

Include this parameter to ignore bending stiffness effects in the shell. Membrane stiffness coefficients are computed normally. Membrane-bending coupling coefficients are set to zero. Diagonal bending stiffness coefficients are set to 1×10^{-6} times the largest diagonal membrane stiffness term. Off-diagonal bending stiffness coefficients are set to zero.

OFFSET

Include this parameter to define the distance (as a fraction of the shell thickness) from the shell midsurface to the reference surface (containing the nodes of the element). This parameter accepts positive or negative values, the labels SPOS or SNEG, or in an Abaqus/Standard analysis the name of a distribution (see "Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual).

Positive values of the offset are in the positive normal direction (see "Shell elements: overview," Section 26.6.1 of the Abaqus Analysis User's Manual). When OFFSET=0.5 (or SPOS), the top surface of the shell is the reference surface. When OFFSET=−0.5 (or SNEG), the bottom surface of the shell is the reference surface. The default is OFFSET=0, which indicates that the middle surface of the shell is the reference surface. This parameter is ignored for continuum shells.

In an Abaqus/Standard analysis a spatially varying offset can be specified by setting OFFSET equal to the name of a distribution. The distribution used to define the shell offset must have a default value. The default offset is used by any shell element assigned to the shell section that is not specifically assigned a value in the distribution.

ORIENTATION

Set this parameter equal to the name of an orientation definition (see "Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) to be used with the section forces and section strains.

POISSON

Include this parameter to define the shell thickness direction behavior.

Set this parameter equal to a nonzero value to cause the thickness direction strain under plane stress conditions to be a linear function of the membrane strains. The value of the POISSON parameter must be between −1.0 and 0.5.

Set POISSON=ELASTIC to automatically select this parameter value based on the initial isotropic elastic part of the material definition.

The default is POISSON=0.5.

SMEAR ALL LAYERS

This parameter is relevant only when the COMPOSITE parameter is used.

Include this parameter to ignore the material layer stacking sequence. Membrane-bending coupling terms are set to zero, and bending stiffness terms are computed as $T^2/12$ times the corresponding membrane stiffness terms, where T is the total thickness of the shell.

STACK DIRECTION

This parameter is relevant only for continuum shells.

Set this parameter equal to 1, 2, 3, or ORIENTATION to define the continuum shell stack or thickness direction. Specify one of the numerical values to select the corresponding isoparametric direction of the element as the stack or thickness direction. The default is STACK DIRECTION=3.

If STACK DIRECTION=ORIENTATION, the ORIENTATION parameter is also required.

To obtain a desired thickness direction, the appropriate numerical value for the STACK DIRECTION parameter depends on the element connectivity. For a mesh-independent specification, use STACK DIRECTION=ORIENTATION. If the orientation assigned to the ORIENTATION parameter is defined with a distribution (“Distribution definition,” Section 2.7.1 of the Abaqus Analysis User’s Manual), STACK DIRECTION=ORIENTATION is not supported.

SYMMETRIC

This parameter is relevant only when the COMPOSITE parameter is used.

Include this parameter if the layers in the composite shell are symmetric about a central core. This parameter cannot be used if a spatially varying thickness or orientation angle is defined on any composite layer using a distribution (“Distribution definition,” Section 2.7.1 of the Abaqus Analysis User’s Manual).

THICKNESS MODULUS

This parameter is relevant only for continuum shells.

Set this parameter equal to an effective thickness modulus. The default effective thickness modulus is twice the initial in-plane shear modulus based on the material definition.

ZERO

If the section is defined by its general stiffness, set this parameter equal to θ^0, the reference temperature for thermal expansion (for example, ZERO=50 means $\theta^0 = 50°$).

This parameter is ignored if the COMPOSITE, the MATERIAL, or the USER parameter is specified.

The following parameters are optional, mutually exclusive, and used only if the section is not defined by its general stiffness on the data lines:

COMPOSITE

Include this parameter to indicate that the shell is composed of layers with different linear elastic material behavior.

MATERIAL

Set this parameter equal to the name of the single linear elastic material of which the shell is made.

USER

This parameter applies only to Abaqus/Standard analyses and cannot be used with continuum shell elements. See "Using a general shell section to define the section behavior," Section 26.6.6 of the Abaqus Analysis User's Manual, for use of this option in linear perturbation analyses.

Include this parameter to indicate that the shell section stiffness is defined in user subroutine **UGENS**.

The following parameters are optional, mutually exclusive, and can be used only in combination with the MATERIAL, the COMPOSITE, or the USER parameter:

NODAL THICKNESS

Include this parameter to indicate that the shell thickness should not be read from the data lines but should be interpolated from the thickness specified at the nodes with the *NODAL THICKNESS option. For composite sections the total thickness is interpolated from the nodes, and the thicknesses of the layers specified on the data lines are scaled proportionally. This parameter is ignored for continuum shells.

SHELL THICKNESS

Set this parameter equal to the name of a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual) to define spatially varying thickness. If this parameter is used for a non-composite section, the thickness on the data line is ignored. For composite sections the total thickness is defined by the distribution, and the thicknesses of the layers specified on the data lines are scaled proportionally. This parameter is ignored for continuum shells.

The distribution used to define shell thickness must have a default value. The default thickness is used by any shell element assigned to the shell section that is not specifically assigned a value in the distribution.

The following optional parameters can be used only in combination with the USER parameter:

I PROPERTIES

Set this parameter equal to the number of integer property values needed as data in user subroutine **UGENS**. The default is I PROPERTIES=0.

PROPERTIES

Set this parameter equal to the number of real (floating point) property values needed as data in user subroutine **UGENS**. The default is PROPERTIES=0.

UNSYMM

Include this parameter if the section stiffness matrices are not symmetric. This parameter will invoke the unsymmetric equation solution capability.

VARIABLES

Set this parameter equal to the number of solution-dependent variables that must be stored for the section. The default is VARIABLES=1.

Optional parameter for use when the MATERIAL, the COMPOSITE, and the USER parameters are omitted:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the scaling moduli, in addition to temperature. If this parameter is omitted, it is assumed that the moduli are constant or depend only on temperature.

Data line if the MATERIAL parameter is included:

First (and only) line:

1. Shell thickness. This value is ignored if the NODAL THICKNESS or SHELL THICKNESS parameter is included.

Data lines if the COMPOSITE parameter is included:

First line:

1. Positive scalar value defining layer thickness or in Abaqus/Standard the name of a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual) that defines spatially varying layer thicknesses. A distribution for composite layer thickness can be used only for conventional shell elements (not continuum shell elements). The layer thickness is modified if the NODAL THICKNESS or SHELL THICKNESS parameter is included.
2. This field is not used (any value given will be ignored). It is included for compatibility with the *SHELL SECTION option.
3. Name of the material forming this layer.
4. Orientation angle, ϕ, or the name of a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual) that defines spatially varying orientation angles. Orientation angles (in degrees) are measured positive counterclockwise relative to the orientation definition given with the ORIENTATION parameter. If the ORIENTATION parameter is not included, ϕ is measured relative to the default shell local directions (see "Conventions," Section 1.2.2 of the Abaqus Analysis User's Manual).

5. Name of the ply. Required only for composite layups defined in Abaqus/CAE.

Repeat this data line as often as necessary to define the layers of the shell. The order of the laminated shell layers with respect to the positive direction of the shell normal is defined by the order of the data lines. If the SYMMETRIC parameter is included, specify only half the layers, from the bottom layer to the midplane.

Data lines to define the shell section directly if the MATERIAL, the COMPOSITE, and the USER parameters are omitted:

First line:

1. Symmetric half of the $[\mathbf{D}]$ matrix in the order D_{11}, D_{12}, D_{22}, D_{13}, D_{23}, D_{33}, D_{14},..., D_{44}, D_{15},..., D_{55},..., D_{16},..., D_{66}

Repeat this data line three times. Enter 21 entries total, 8 per line.

Second line (optional):

1. $\{\mathbf{F}\}$ vector (F_1, F_2, ..., F_6).

Third line (optional):

1. Y, scaling modulus for $[\mathbf{D}]$. The default is 1.0.
2. α, scaling modulus for $[\mathbf{F}]$. The default is 0.0.
3. θ, temperature for these values of Y and α.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define Y and α as functions of temperature and other predefined field variables.

Data lines to define spatially varying shell section stiffness with a distribution if the MATERIAL, the COMPOSITE, and the USER parameters are omitted:

First line:

1. Distribution name. The data in the distribution define the symmetric half of the $[\mathbf{D}]$ matrix.

Second line (optional):

1. $\{\mathbf{F}\}$ vector (F_1, F_2, ..., F_6).

Third line (optional):

1. Y, scaling modulus for $[\mathbf{D}]$. The default is 1.0.
2. α, scaling modulus for $[\mathbf{F}]$. The default is 0.0.
3. θ, temperature for these values of Y and α.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define Y and α as functions of temperature and other predefined field variables.

Data lines if the USER parameter is included:

First line:

1. Shell thickness. This value is ignored if the NODAL THICKNESS or SHELL THICKNESS parameters are included.

Second line:

1. Enter the data to be used as properties in user subroutine **UGENS**. Enter all floating point values first, followed immediately by the integer values.

Repeat this data line as often as necessary to define the properties required in **UGENS**. Enter eight values per line for both real and integer values.

18.15 *SHELL SECTION: Specify a shell cross-section.

This option is used to specify a shell cross-section.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- "Shell elements: overview," Section 26.6.1 of the Abaqus Analysis User's Manual
- "Using a shell section integrated during the analysis to define the section behavior," Section 26.6.5 of the Abaqus Analysis User's Manual

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the shell elements for which the section behavior is being defined.

Required, mutually exclusive parameters:

COMPOSITE

Include this parameter if the shell is made up of several layers of material.

MATERIAL

Set this parameter equal to the name of the material of which the shell is made.

Optional parameters:

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see "Section controls," Section 24.1.4 of the Abaqus Analysis User's Manual) to be used to specify the second-order accurate element formulation option, a nondefault hourglass control formulation option, or scale factors.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition to be used to specify the enhanced hourglass control formulation (see "Section controls,"

Section 24.1.4 of the Abaqus Analysis User's Manual) or to be used in a subsequent Abaqus/Explicit import analysis.

DENSITY

Set this parameter equal to a mass per unit surface area of the shell.

If this parameter is used, the mass of the shell includes a contribution from this parameter in addition to any contribution from the material definition.

LAYUP

This parameter is relevant only when the COMPOSITE parameter is used.

Set this parameter equal to the name of a composite layup (see Chapter 22, "Composite layups," of the Abaqus/CAE User's Manual). Abaqus/CAE uses this name to identify the composite layup that contains the shell section.

NODAL THICKNESS

Include this parameter to indicate that the shell thickness should not be read from the data lines but should be interpolated from the thickness specified at the nodes with the *NODAL THICKNESS option. For composite sections the total thickness is interpolated from the nodes and the thicknesses of the layers specified on the data lines are scaled proportionally. This parameter is ignored for continuum shells.

The NODAL THICKNESS and SHELL THICKNESS parameters are mutually exclusive.

OFFSET

Include this parameter to define the distance (as a fraction of the shell thickness) from the shell midsurface to the reference surface (containing the nodes of the element). This parameter accepts positive or negative values, the labels SPOS or SNEG, or in an Abaqus/Standard analysis the name of a distribution. See "Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual.

Positive values of the offset are in the positive normal direction (see "Shell elements: overview," Section 26.6.1 of the Abaqus Analysis User's Manual). When OFFSET=0.5 (or SPOS), the top surface of the shell is the reference surface. When OFFSET=−0.5 (or SNEG), the bottom surface of the shell is the reference surface. The default is OFFSET=0, which indicates that the middle surface of the shell is the reference surface. This parameter is ignored for continuum shells.

In an Abaqus/Standard analysis a spatially varying offset can be specified by setting OFFSET equal to the name of a distribution. The distribution used to define the shell offset must have a default value. The default offset is used by any shell element assigned to the shell section that is not specifically assigned a value in the distribution.

ORIENTATION

Set this parameter equal to the name of an orientation definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) to be used with material calculations in this definition of shell section behavior. This orientation will be used for material calculations and stress output in the individual layers, for the section forces output, and for the transverse shear stiffness.

It is possible to use a different orientation definition for material calculations in the individual layers of a composite shell by referencing an orientation definition or giving an orientation angle

(in degrees, positive counterclockwise relative to the shell local directions) on each layer definition data line. Any layer definition line that does not have an orientation reference or an angle specified will use the orientation defined here. While an orientation defined with a distribution can be used to specify the overall orientation for the shell section, an orientation defined with a distribution cannot be specified on the layers of a composite shell.

POISSON

Include this parameter to define the shell thickness direction behavior.

Set this parameter equal to a nonzero value to cause the thickness direction strain under plane stress conditions to be a linear function of the membrane strains. The value of the POISSON parameter must be between −1.0 and 0.5.

Set POISSON=ELASTIC to automatically select this parameter value based on the initial elastic part of the material definition.

Set POISSON=MATERIAL in an Abaqus/Explicit analysis to cause the thickness direction strain under plane stress conditions to be a function of the membrane strains and the in-plane material properties.

In Abaqus/Standard the default is POISSON=0.5; in Abaqus/Explicit the default is POISSON=MATERIAL.

SECTION INTEGRATION

Set SECTION INTEGRATION=SIMPSON (default) to use Simpson's rule for the shell section integration.

Set SECTION INTEGRATION=GAUSS to use Gauss quadrature for the shell section integration. Gauss quadrature cannot be used for heat transfer or thermally coupled shell elements.

SHELL THICKNESS

Set this parameter equal to the name of a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual) to define spatially varying thickness. If this parameter is used for a non-composite section the thickness on the data line is ignored. For composite sections the total thickness is defined by the distribution and the thicknesses of the layers specified on the data lines are scaled proportionally. This parameter is ignored for continuum shells.

The distribution used to define shell thickness must have a default value. The default thickness is used by any shell element assigned to the shell section that is not specifically assigned a value in the distribution.

The NODAL THICKNESS and SHELL THICKNESS parameters are mutually exclusive.

STACK DIRECTION

This parameter is relevant only for continuum shells.

Set this parameter equal to 1, 2, 3, or ORIENTATION to define the continuum shell stack or thickness direction. Specify one of the numerical values to select the corresponding isoparametric direction of the element as the stack or thickness direction. The default is STACK DIRECTION=3.

If STACK DIRECTION=ORIENTATION, the ORIENTATION parameter is also required.

To obtain a desired thickness direction, the appropriate numerical value for the STACK DIRECTION parameter depends on the element connectivity. For a mesh-independent

specification, use STACK DIRECTION=ORIENTATION. If the orientation assigned to the ORIENTATION parameter is defined with a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual), STACK DIRECTION=ORIENTATION is not supported.

SYMMETRIC

This parameter is relevant only when the COMPOSITE parameter is used.

Include this parameter if the layers in the composite shell are symmetric about a central core. This parameter cannot be used if a spatially varying thickness or orientation angle is defined on any composite layer using a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual).

TEMPERATURE

Use this parameter to select the mode of temperature and field variable input used on the *FIELD, the *INITIAL CONDITIONS, or the *TEMPERATURE options.

Omit the TEMPERATURE parameter to define the predefined field by its magnitude on the reference surface of the shell and its gradient through the thickness.

Set TEMPERATURE=*n*, where *n* is the number of predefined field variable points in the shell or in each layer if the COMPOSITE parameter is used, to define the predefined field at *n* equally spaced points through each layer of the shell section.

In a heat transfer analysis step or a coupled temperature-displacement analysis step when the *FIELD option is used to specify values of predefined field variables, the TEMPERATURE parameter is required to specify the number of field variable points in the shell. The number of temperature points in the shell with temperature degrees of freedom is defined by the number of integration points specified on the data lines.

THICKNESS MODULUS

This parameter is relevant only for continuum shells.

Set this parameter equal to an effective thickness modulus. The default effective thickness modulus is twice the initial in-plane shear modulus based on the material definition.

Data line to define a homogeneous shell (the MATERIAL parameter is included):

First (and only) line:

1. Shell thickness. This value is ignored if the NODAL THICKNESS or SHELL THICKNESS parameters are included.
2. Number of integration points to be used through the shell section. The default is five points if Simpson's rule is used and three points if Gauss quadrature is used. The number of integration points must be an odd number for Simpson's rule and is equal to the number of temperature degrees of freedom at a node of the element if this section is associated with heat transfer or coupled temperature-displacement elements. The maximum number of points for Simpson's rule is 99, and in the case of heat transfer or coupled temperature-displacement elements it is 19. This number must be at least 2 and less than or equal to 15 for Gauss quadrature. For

Simpson's rule it must be at least 3, except in a pure heat transfer analysis, where the number of integration points can be 1 for a constant temperature through the shell thickness.

Data lines to define a composite shell (the COMPOSITE parameter is included):

First line:

1. Positive scalar value defining layer thickness or in Abaqus/Standard the name of a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual) that defines spatially varying layer thicknesses. A distribution for composite layer thickness can be used only for conventional shell elements (not continuum shell elements). The layer thickness is modified if the NODAL THICKNESS or SHELL THICKNESS parameter is included.
2. Number of integration points to be used through the layer. The default is three points if Simpson's rule is used and two points if Gauss quadrature is used. The number of integration points must be an odd number for Simpson's rule, and it determines the number of temperature degrees of freedom at a node of the element if this section is associated with heat transfer or coupled temperature-displacement elements. The maximum number of points for Simpson's rule is 99, and in the case of heat transfer or coupled temperature-displacement elements it is 19. This number must be less than or equal to 15 for Gauss quadrature.
3. Name of the material forming this layer.
4. Name of the orientation to be used with this layer, an orientation angle, ϕ, or the name of a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual) that defines spatially varying orientation angles. If the name of an orientation is used, the orientation cannot be defined with distributions. Orientation angles (in degrees) are measured positive counterclockwise relative to the orientation definition given with the ORIENTATION parameter. If the ORIENTATION parameter is not included, ϕ is measured relative to the default shell local directions (see "Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual).
5. Name of the ply. Required only for composite layups defined in Abaqus/CAE.

Repeat this data line as often as necessary. Use one data line for each layer of the shell. The order of the laminated shell layers with respect to the positive direction of the shell normal is defined by the order of the data lines. If the SYMMETRIC parameter is included, specify only half the layers, from the bottom layer to the midplane.

18.16 *SHELL TO SOLID COUPLING: Define a surface-based coupling between a shell edge and a solid face.

This surface-based option allows for a transition from shell element modeling to solid element modeling in a three-dimensional analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Manual
- "Node-based surface definition," Section 2.3.3 of the Abaqus Analysis User's Manual
- "Shell-to-solid coupling," Section 31.3.3 of the Abaqus Analysis User's Manual

Required parameter:

CONSTRAINT NAME

Set this parameter equal to a label that will be used to refer to this constraint.

Optional parameters:

INFLUENCE DISTANCE

Set this parameter equal to the perpendicular distance from the edge-based surface within which all nodes or element facets on the solid surface (depending on the solid surface type) must lie to be included in the coupling constraint. The default value is half the thickness of the shell that was used to define the edge-based surface.

POSITION TOLERANCE

Set this parameter equal to the distance within which nodes on the edge-based surface must lie from the solid surface to be included in the coupling definition. The default tolerance is 5% of the length of a typical facet on the shell edge.

Data lines to define the surfaces forming the coupling definitions:

First line:

1. The edge-based surface name.

2. The solid surface name.

Repeat this data line as often as necessary to define all the surfaces forming the coupling definition. Each data line defines a pair of surfaces that will be coupled.

18.17 *SIMPEDANCE: Define impedances of acoustic surfaces.

This option is used to provide surface impedance information or nonreflecting boundaries for acoustic and coupled acoustic-structural analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Manual
- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *IMPEDANCE
- *IMPEDANCE PROPERTY

Required, mutually exclusive parameters:

PROPERTY

Set this parameter equal to the name of the *IMPEDANCE PROPERTY option defining the table of impedance values to be used.

NONREFLECTING

Set NONREFLECTING=PLANAR (default) to specify the impedance corresponding to that of a normal incidence plane wave.

Set NONREFLECTING=IMPROVED to specify the impedance corresponding to that of a plane wave at an arbitrary angle of incidence. This parameter can be used only for transient dynamics.

Set NONREFLECTING=CIRCULAR to specify a radiation condition appropriate for a circular boundary in two dimensions or a right circular cylinder in three dimensions.

Set NONREFLECTING=SPHERICAL to specify a radiation condition appropriate for a spherical boundary.

Set NONREFLECTING=ELLIPTICAL to specify a radiation condition appropriate for an elliptical boundary in two dimensions or a right elliptical cylinder in three dimensions.

Set NONREFLECTING=PROLATE SPHEROIDAL to specify a radiation condition appropriate for a prolate spheroidal boundary.

Optional parameter:

OP

Set OP=MOD (default) to modify existing impedances or to define additional impedances.

Set OP=NEW if all existing impedances applied to the model should be removed. To remove only selected impedances, use OP=NEW and respecify all impedances that are to be retained.

Data line to define an impedance for PROPERTY, NONREFLECTING=PLANAR, or NONREFLECTING=IMPROVED:

First (and only) line:

1. Surface name.

Data line to define an absorbing boundary impedance for NONREFLECTING=CIRCULAR or NONREFLECTING=SPHERICAL:

First (and only) line:

1. Surface name.
2. r_1, the radius of the circle or sphere defining the absorbing boundary surface.

Data line to define an absorbing boundary impedance for NONREFLECTING=ELLIPTICAL or NONREFLECTING=PROLATE SPHEROIDAL:

First (and only) line:

1. Surface name.
2. The semimajor axis, *a*, of the ellipse or prolate spheroid defining the surface. *a* is 1/2 of the maximum distance between two points on the ellipse or spheroid, analogous to the radius of a circle or sphere.
3. The eccentricity, ϵ, of the ellipse or prolate spheroid. The eccentricity is the square root of one minus the square of the ratio of the minor axis, *b*, to the major axis, *a*: $\epsilon = \sqrt{1 - (b/a)^2}$.
4. Global *X*-coordinate of the center of the radiating surface.
5. Global *Y*-coordinate of the center of the radiating surface.
6. Global *Z*-coordinate of the center of the radiating surface.
7. *X*-component of the direction cosine of the major axis of the radiating surface. The components of this vector need not be normalized to unit magnitude.
8. *Y*-component of the direction cosine of the major axis of the radiating surface.
9. *Z*-component of the direction cosine of the major axis of the radiating surface.

18.18 *SIMPLE SHEAR TEST DATA: Used to provide simple shear test data.

This option is used to provide simple shear test data. It can be used only in conjunction with the *HYPERFOAM option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Hyperelastic behavior in elastomeric foams," Section 19.5.2 of the Abaqus Analysis User's Manual
- *HYPERFOAM

There are no parameters associated with this option.

Data lines to specify simple shear test data for a hyperfoam:

First line:

1. Nominal shear stress, T_S.
2. Nominal shear strain, γ.
3. Nominal transverse stress, T_T (normal to edge with shear stress). This stress value is optional but strongly recommended. If given, a more accurate material response will be obtained.

Repeat this data line as often as necessary to give the stress-strain data.

18.19 *SLIDE LINE: Specify slide line surfaces on which deformable structures may interact.

This option is relevant only for slide line and tube-to-tube contact elements. It is used to define the slide line and to specify which set of contact elements interacts with it.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Assembly

Abaqus/CAE: Unsupported; the Interaction module uses surface-based contact.

References:

- "Tube-to-tube contact elements," Section 36.3.1 of the Abaqus Analysis User's Manual
- "Slide line contact elements," Section 36.4.1 of the Abaqus Analysis User's Manual

Required parameters:

ELSET

Set this parameter equal to the name of the element set containing the slide line or tube-to-tube contact elements that interact with the slide line being defined.

TYPE

Set TYPE=LINEAR to define a slide line made up of linear segments. This parameter should be used when first-order elements make up the model.

Set TYPE=PARABOLIC to define a slide line made up of parabolic segments. This parameter should be used when second-order elements make up the model. In this case the slide line should consist of an odd number of nodes, where the odd nodes on the slide line are associated with the ends of parabolic segments.

Optional parameters:

EXTENSION ZONE

Set this parameter equal to a fraction of the end segment length by which either end of an open slide line is to be extended to avoid numerical round off errors associated with contact modeling. The value given must lie between 0.0 and 0.2. The default value is 0.1.

GENERATE

Include this parameter to allow incremental generation of node numbers along a slide line.

SMOOTH

Set this parameter equal to the smoothing fraction, *f*, to round discontinuities between line segments of a slide line. The default is 0. The limit is $f < 0.5$.

Data lines if the GENERATE parameter is omitted:

First line:

1. First node number on this slide line.
2. Second node number on this slide line.
3. Third node number on this slide line.
4. Etc.

Repeat this data line as often as necessary to specify the nodes forming the slide line. Enter up to 16 integer values per line.

Data lines if the GENERATE parameter is included:

First line:

1. First node number of the series.
2. Last node number of the series.
3. Increment between node numbers.

Repeat this data line as often as necessary to specify the nodes forming the slide line.

18.20 *SLOAD: Apply loads to a substructure.

This option is used to activate a substructure load case defined by the *SUBSTRUCTURE LOAD CASE option.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Manual
- *SUBSTRUCTURE LOAD CASE

Optional parameters:

AMPLITUDE

Set this parameter equal to the name given to an amplitude defined by the *AMPLITUDE option ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual). This amplitude defines the time variation of the load case (*SUBSTRUCTURE LOAD CASE) magnitude throughout the step. If this parameter is omitted, the default amplitude is that defined in "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual, for the particular procedure used in the step.

OP

Set OP=MOD (default) to modify or add to the currently active *SLOADs.

Set OP=NEW if all existing *SLOADs applied to the model should be removed and new ones possibly defined.

Data lines to define the loading:

First line:

1. Element number or element set name. The elements must be substructures.
2. Load case name (as defined on the associated *SUBSTRUCTURE LOAD CASE option).
3. Load magnitude. This magnitude is a multiplier for the load case magnitude defined under the *SUBSTRUCTURE LOAD CASE option. It can be adjusted throughout the step by the amplitude specification in the usual way.

Repeat this data line as often as necessary to define the loadings on the substructures.

18.21 *SOILS: Effective stress analysis for fluid-filled porous media.

This option is used to specify transient (consolidation) or steady-state response analysis of partially or fully saturated fluid-filled porous media.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Coupled pore fluid diffusion and stress analysis," Section 6.8.1 of the Abaqus Analysis User's Manual
- "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual

Optional parameters:

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified via the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual).

CETOL

This parameter will invoke automatic time incrementation. If the UTOL, DELTMX, and CETOL parameters are omitted, fixed time increments will be used.

This parameter is meaningful only when the material response includes time-dependent creep behavior; CETOL controls the accuracy of the creep integration. Set this parameter equal to the maximum allowable difference in the creep strain increment calculated from the creep strain rates at the beginning and at the end of the increment (see "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual).

The tolerance can be calculated by choosing an acceptable stress error tolerance and dividing by a typical elastic modulus.

CONSOLIDATION

Include this parameter to choose transient (consolidation) analysis. Omit this parameter to choose steady-state analysis.

CONTINUE

Set CONTINUE=NO (default) to specify that this step will not carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step will carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

DELTMX

This parameter will invoke automatic time incrementation. Set this parameter equal to the maximum temperature change allowed within an increment. Abaqus/Standard will restrict the time step to ensure that this value is not exceeded at any node during any increment of the step. If this parameter, the CETOL parameter, and the UTOL parameter are all omitted in a transient analysis, fixed time increments will be used, with a constant time increment equal to the initial time increment.

END

This parameter is meaningful only for transient analysis. Set END=PERIOD (default) to analyze a specified time period. Set END=SS to end the step when steady state is reached.

CREEP

Set CREEP=NONE to specify that there is no creep or viscoelastic response occurring during this step even if creep or viscoelastic material properties have been defined.

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see “Solving nonlinear problems,” Section 7.1.1 of the Abaqus Analysis User’s Manual) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus/Standard is not suitable. This parameter must be used in conjunction with the STABILIZE and CONSOLIDATION parameters and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

HEAT

This parameter is relevant if there are regions in the model that use coupled temperature–pore pressure elements; it specifies whether heat transfer effects are to be modeled in these regions.

This parameter is not relevant if only coupled pore pressure–displacement elements are used in a model.

Set HEAT=YES (default) to specify that heat transfer effects are to be modeled in these regions. In this case Abaqus/Standard solves the heat transfer equation in conjunction with the mechanical equilibrium and the fluid flow continuity equations.

Set HEAT=NO to specify that heat transfer will not be modeled in these regions.

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is 2×10^{-4} and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL =0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be overridden by the damping factor.

This parameter can be used only in conjunction with the CONSOLIDATION parameter.

UTOL

This parameter will invoke automatic time incrementation. If the UTOL, DELTMX, and the CETOL parameters are omitted, fixed time increments will be used.

Set this parameter equal to the maximum pore pressure change permitted in any increment (in pressure units) in a transient consolidation analysis. Abaqus/Standard will restrict the time step to ensure that this value will not be exceeded at any node (except nodes with boundary conditions) during any increment of the analysis.

Set this value equal to any nonzero value in a steady-state analysis (to activate automatic time incrementation).

Data line to define incrementation for a soils analysis:

First (and only) line:

1. Initial time increment. This value will be modified as required if the automatic time stepping scheme is used; it will be used as a constant time increment otherwise.
2. Time period. If END=SS is chosen, the step ends after this time period or when steady-state conditions are reached, whichever comes first.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed. Only used for automatic time incrementation.
4. Maximum time increment allowed. If this value is not specified, the upper limit is the total time period. Only used for automatic time incrementation.
5. The rate of change of pore pressure with time, used to define steady state: only needed if END=SS is chosen. When all nodal wetting liquid pressures are changing at rates below this value, the solution terminates.

6. The rate of change of temperature with time, used to define steady state: only needed if END=SS is chosen. When all nodal temperatures are changing at rates below this value, the solution terminates.

18.22 *SOLID SECTION: Specify element properties for solid, infinite, acoustic, and truss elements.

This option is used to define properties of solid (continuum) elements, infinite elements, acoustic finite and infinite elements, and truss elements.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- "Solid (continuum) elements," Section 25.1.1 of the Abaqus Analysis User's Manual
- "Infinite elements," Section 25.3.1 of the Abaqus Analysis User's Manual
- "Truss elements," Section 26.2.1 of the Abaqus Analysis User's Manual

Required parameters:

COMPOSITE

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only with three-dimensional brick solid elements that have only displacement degrees of freedom. Include this parameter if the solid is made up of several layers of material.

The COMPOSITE and MATERIAL parameters are mutually exclusive.

ELSET

Set this parameter equal to the name of the element set containing the elements for which the material behavior is being defined.

MATERIAL

Set this parameter equal to the name of the material to be used with these elements.

The COMPOSITE and MATERIAL parameters are mutually exclusive.

REF NODE

This parameter is required only for generalized plane strain elements and acoustic infinite elements; it is ignored for all other element types.

Set this parameter equal to either the node number of the reference node or the name of a node set containing the reference node. If the name of a node set is chosen, the node set must contain exactly one node.

Required parameter for anisotropic materials; optional parameter for isotropic materials:

ORIENTATION

Set this parameter equal to the name of an orientation definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) to be used to define a local coordinate system for material calculations in the elements in this set. This parameter is required when the material is anisotropic.

For a composite solid this orientation, together with the orientation angle specified on the layer data lines, can also be used to define the material orientations in the individual layers. Alternatively, a material orientation can be specified by referencing an orientation definition on each layer data line. In this case the reference given on the ORIENTATION parameter is ignored. Any layer definition line that does not have an orientation reference or an angle specified will use the section orientation defined on the keyword line.

Optional parameters:

CONTROLS

In an Abaqus/Explicit analysis, set this parameter equal to the name of a section controls definition (see "Section controls," Section 24.1.4 of the Abaqus Analysis User's Manual) to be used to specify a nondefault hourglass control formulation option or scale factor. The *SECTION CONTROLS option can be used to select the hourglass control and order of accuracy of the formulation for two- and three-dimensional solid elements and to select the kinematic formulation for 8-node brick elements.

In an Abaqus/Standard analysis, set this parameter equal to the name of a section controls definition (see "Section controls," Section 24.1.4 of the Abaqus Analysis User's Manual) to be used to specify the enhanced hourglass control formulation or to be used in a subsequent Abaqus/Explicit import analysis.

LAYUP

This parameter is relevant only when the COMPOSITE parameter is used.

Set this parameter equal to the name of a composite layup (see Chapter 22, "Composite layups," of the Abaqus/CAE User's Manual). Abaqus/CAE uses this name to identify the composite layup that contains the solid section.

ORDER

This parameter can be used only with acoustic infinite elements in Abaqus/Explicit. It defines the number of ninth-order polynomials that will be used to resolve the variation of the acoustic field in the infinite direction. Set this parameter equal to N to indicate that the first N members of the set of ninth-order polynomials are to be used. The default is ORDER=10, which is the value always used in Abaqus/Standard.

STACK DIRECTION

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only with composite elements. It defines the stacking direction with respect to a pair of element faces. Set this parameter equal to 1, 2, or 3. The default is STACK DIRECTION=3.

SYMMETRIC

This parameter is relevant only when the COMPOSITE parameter is used.

Include this parameter if the layers in the composite shell are symmetric about a central core. This parameter cannot be used if spatially varying orientation angles are defined on any composite layer using distributions ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual).

Data line to define homogeneous solid elements, infinite elements, acoustic elements, or truss elements:

First (and only) line:

1. Enter any attribute values required. The default for the first attribute is 1.0. See the description in Part VI, "Elements," of the Abaqus Analysis User's Manual of the element type being used for a definition of the data required.

Data lines to define a composite solid:

First line:

1. Layer thickness. The layer thickness will be adjusted such that the sum of the layer thicknesses corresponds to the element length in the stack direction.
2. Number of integration points to be used through the layer. This number must be an odd number. The default is one integration point.
3. Name of the material forming this layer.
4. Name of the orientation to be used with this layer, an orientation angle, ϕ, or in Abaqus/Standard the name of a distribution ("Distribution definition," Section 2.7.1 of the Abaqus Analysis User's Manual) that defines spatially varying orientation angles. If the name of an orientation is used, the orientation cannot be defined with distributions. Orientation angles (in degrees) are measured positive counterclockwise relative to the local direction, which must be defined on the *ORIENTATION definition.
5. Name of the ply. Required only for composite layups defined in Abaqus/CAE.

Repeat this data line as often as necessary to define the properties for each layer of the composite solid. If the SYMMETRIC parameter is included, specify only half the layers, from the bottom layer to the midplane.

18.23 *SOLUBILITY: Specify solubility.

This option is used to define the solubility for a material diffusing through a base material. It must be used in conjunction with the *DIFFUSIVITY option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Solubility," Section 23.5.2 of the Abaqus Analysis User's Manual
- *DIFFUSIVITY

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the solubility. If this parameter is omitted, it is assumed that the solubility is constant or that it depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define solubility:

First line:

1. Solubility. (Units of $PLF^{-1/2}$.)
2. Temperature, θ.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the solubility as a function of temperature and other predefined field variables.

18.24 *SOLUTION TECHNIQUE: Specify alternative solution methods.

This option is used to specify the quasi-Newton method instead of the standard Newton method for solving nonlinear equations, to specify a separated solution scheme for *COUPLED TEMPERATURE-DISPLACEMENT and *COUPLED THERMAL-ELECTRICAL procedures, or to specify that contact iterations should be executed instead of regular severe discontinuity iterations.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Fully coupled thermal-stress analysis," Section 6.5.4 of the Abaqus Analysis User's Manual
- "Coupled thermal-electrical analysis," Section 6.7.2 of the Abaqus Analysis User's Manual
- "Contact iterations," Section 7.1.2 of the Abaqus Analysis User's Manual
- "Convergence criteria for nonlinear problems," Section 7.2.3 of the Abaqus Analysis User's Manual

Required parameter:

TYPE

Set TYPE=QUASI-NEWTON to specify a quasi-Newton solution method.

Set TYPE=SEPARATED to specify that linearized equations for the individual fields in a fully coupled procedure are to be decoupled and solved separately for each field. This option can be specified only with the *COUPLED TEMPERATURE-DISPLACEMENT and *COUPLED THERMAL-ELECTRICAL procedures.

Set TYPE=CONTACT ITERATIONS to specify that contact iterations should be executed instead of regular severe discontinuity iterations.

Optional parameter:

REFORM KERNEL

This parameter can be used only with TYPE=QUASI-NEWTON. Set this parameter equal to the number of quasi-Newton iterations allowed before the kernel matrix is reformed. The default is REFORM KERNEL=8.

Data line for TYPE=CONTACT ITERATIONS:

First (and only) line:

1. Correction factor on the maximum number of right-hand-side solutions during any contact iteration. The default is 1. The actual number of allowed right-hand-side solutions with the correction factor accounted for is printed in the message file if *PRINT, CONTACT=YES is specified.
2. Maximum number of contact iterations allowed before new global matrix assemblage and factorization. The default is 30.

18.25 *SOLVER CONTROLS: Specify controls for the iterative and direct linear solvers.

This option is used to set the control parameters for both the direct and iterative linear equation solvers.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Direct linear equation solver," Section 6.1.4 of the Abaqus Analysis User's Manual
- "Iterative linear equation solver," Section 6.1.5 of the Abaqus Analysis User's Manual

Optional parameters:

CONSTRAINT OPTIMIZATION

Include this parameter to optimize the handling of hard contact and compressibility constraints associated with hybrid elements.

RESET

Include this parameter to reset all of the iterative solver controls to their default values. The option should have no data lines when this parameter is used.

If this parameter is omitted, only the iterative solver specified controls will be changed in the current step; the other controls will remain at their settings from previous steps.

Data line to define control parameters for the iterative solver:

First (and only) line:

1. Relative tolerance for convergence. The default is 10^{-3} for all general procedures. For linear perturbation procedures, the default is 10^{-6}.
2. Maximum number of iterations allowed. The default is 300.

18.26 *SORPTION: Define absorption and exsorption behavior.

This option is used to define absorption and exsorption behaviors of a partially saturated porous medium in the analysis of coupled wetting liquid flow and porous medium stress.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Sorption," Section 23.7.4 of the Abaqus Analysis User's Manual

Optional parameters:

LAW

Set LAW=LOG to define the absorption or exsorption behavior by the analytical logarithmic form.

Set LAW=TABULAR (default) to define the absorption or exsorption behavior in tabulated form.

TYPE

Set TYPE=ABSORPTION (default) to define the absorption behavior.

Set TYPE=EXSORPTION to define the exsorption behavior (this must be a repeated use of the option for the same material).

Set TYPE=SCANNING to define the scanning line (this must be a repeated use of the option for the same material).

Data lines for TYPE=ABSORPTION or TYPE=EXSORPTION and LAW=TABULAR:

First line:

1. Pore pressure, u_w, with the condition $u_w \leq 0.0$. (Units of FL^{-2}.)
2. Saturation, s. This value must lie in the range $0.01 \leq s \leq 1.0$.

Repeat this data line as often as necessary to define the relationship between u_w and s from $s = 0^+$ to $s = 1.0$ in increasing values of s. At least two data lines must be specified.

Data line for TYPE=ABSORPTION or TYPE=EXSORPTION and LAW=LOG:

First (and only) line:

1. A. This value must be positive. (Dimensionless.)
2. B. This value must be positive. (Units of L^2F^{-1}.)
3. s_0. This value must lie in the range $0.01 \leq s_0 < s_1 < 0.9$. The default is 0.01.
4. s_1. This value must lie in the range $0.01 \leq s_0 < s_1 < 0.9$. The default is 0.01 plus a very small positive number (since s_1 cannot be equal to s_0).

Data line for TYPE=SCANNING:

First (and only) line:

1. Slope of the scanning line, $(du_w/ds)\big|_s$. This slope must be positive and larger than the slope of any segment of the absorption or exsorption behavior definitions.

18.27 *SPECIFIC HEAT: Define specific heat.

This option is used to specify a material's specific heat.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Specific heat," Section 23.2.3 of the Abaqus Analysis User's Manual

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variables included in the definition of specific heat. If this parameter is omitted, it is assumed that the specific heat is constant or depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

PORE FLUID

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the specific heat of the pore fluid in a porous medium is being defined.

Data lines to specify a material's specific heat:

First line:

1. Specific heat per unit mass. (Units of $\mathrm{JM}^{-1}\theta^{-1}$.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the specific heat as a function of temperature and other predefined field variables.

18.28 *SPECTRUM: Define a response spectrum.

This option is used to define a spectrum to be used in a *RESPONSE SPECTRUM analysis.

Product: Abaqus/Standard

Type: Model data

Level: Model

References:

- "Response spectrum analysis," Section 6.3.10 of the Abaqus Analysis User's Manual
- *RESPONSE SPECTRUM

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the spectrum. This label is used to cross-reference a spectrum on the *RESPONSE SPECTRUM option.

Optional parameters:

G

Set this parameter equal to the value of the acceleration of gravity. This parameter is used only when TYPE=G is specified.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

TYPE

Set TYPE=DISPLACEMENT, TYPE=VELOCITY, or TYPE=ACCELERATION (default) to define the units in which the spectrum is defined.

Set TYPE=G to enter an acceleration spectrum in *g*-units. The parameter G then defines the gravitational acceleration.

Date lines to define a spectrum:

First line:

1. Magnitude of the spectrum.

2. Frequency, in cycles per time, at which this magnitude is used.
3. Associated damping, given as ratio of critical damping.

Repeat this data line as often as necessary to define the spectrum at all frequencies at each damping value.

18.29 *SPRING: Define spring behavior.

This option is used to define the spring behavior for spring elements. It is also used to assign a structural damping factor to form the imaginary part of the spring stiffness matrix. The imaginary stiffness represents an element-level damping contribution to the frequency domain dynamic equations and to the time domain mode-based dynamic analyses that support nondiagonal damping (see "Modal dynamic analysis," Section 2.5.5 of the Abaqus Theory Manual).

In Abaqus/Standard analyses it is also used to define the spring behavior for ITS and JOINTC elements. If the *SPRING option is being used to define part of the behavior of ITS or JOINTC elements, it must be used in conjunction with the *ITS or *JOINT options and the ELSET and ORIENTATION parameters should not be used.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Property module and Interaction module; supported only for linear behavior independent of field variables. For nonlinear behavior or to include field variables, model connectors in the Interaction module.

References:

- "Springs," Section 29.1.1 of the Abaqus Analysis User's Manual
- "Flexible joint element," Section 29.3.1 of the Abaqus Analysis User's Manual
- "Tube support elements," Section 29.9.1 of the Abaqus Analysis User's Manual

Required parameter if the behavior of spring elements is being defined:

ELSET

Set this parameter equal to the name of the element set containing the spring elements for which this behavior is being defined.

Optional parameters:

COMPLEX STIFFNESS

This parameter is relevant in direct-solution and subspace-based steady-state analyses and in mode-based analyses with *FREQUENCY, DAMPING PROJECTION=ON in Abaqus/Standard that support nondiagonal damping.

Include this parameter to define both the real and imaginary parts of the stiffness. The imaginary part represents structural damping.

If this parameter is omitted, the default is real stiffness only.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the real spring stiffness data, in addition to temperature. If this parameter is omitted, it is assumed that the spring stiffness is independent of field variables. See "Using the DEPENDENCIES parameters to define field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

NONLINEAR

Include this parameter to define nonlinear spring behavior. Omit this parameter to define linear spring behavior.

ORIENTATION

This parameter applies only to Abaqus/Standard analyses.

If the option is being used to define the behavior of SPRING1 or SPRING2 elements, this parameter can be used to refer to an orientation definition so that the spring is acting in a local system. Set this parameter equal to the name of the *ORIENTATION definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual).

RTOL

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the tolerance to be used for regularizing the material data. The default is RTOL=0.03. See "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for a discussion of data regularization.

Data lines to define linear spring behavior for SPRINGA or ITS elements:

First line:

1. Enter a blank line.

Second line:

1. Spring stiffness (force per relative displacement).
2. Frequency (in cycles per time). Applicable only for *STEADY STATE DYNAMICS, DIRECT; *STEADY STATE DYNAMICS, SUBSPACE PROJECTION; and *STEADY STATE DYNAMICS and *MODAL DYNAMIC analyses that support nondiagonal damping.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the spring stiffness as a function of frequency, temperature, and other predefined field variables.

Data lines to define nonlinear spring behavior for SPRINGA or ITS elements:

First line:

1. Enter a blank line.

Second line:

1. Force.
2. Relative displacement.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the spring stiffness as a function of temperature and other predefined field variables.

Data lines to define linear spring behavior for SPRING1, SPRING2, or JOINTC elements:

First line:

1. Degree of freedom with which the springs are associated at their first nodes or, for JOINTC elements, the degree of freedom for which the spring behavior is being defined.
2. For SPRING2 elements give the degree of freedom with which the springs are associated at their second nodes.

If the ORIENTATION parameter is included on the *SPRING option when defining spring elements or on the *JOINT option when defining joint elements, the degrees of freedom specified here are in the local system defined by the *ORIENTATION option referenced.

Second line:

1. Real spring stiffness (force per relative displacement).
2. Frequency (in cycles per time). Applicable only for *STEADY STATE DYNAMICS, DIRECT; *STEADY STATE DYNAMICS, SUBSPACE PROJECTION; and *STEADY STATE DYNAMICS and *MODAL DYNAMIC analyses that support nondiagonal damping.
3. Temperature.

4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the spring stiffness as a function of frequency, temperature, and other predefined field variables.

Data lines to define nonlinear spring behavior for SPRING1, SPRING2, or JOINTC elements:

First line:

1. Degree of freedom with which the springs are associated at their first nodes or, for JOINTC elements, the degree of freedom for which the spring behavior is being defined.
2. For SPRING2 elements give the degree of freedom with which the springs are associated at their second nodes.

If the ORIENTATION parameter is included on the *SPRING option when defining spring elements or on the *JOINT option when defining joint elements, the degrees of freedom specified here are in the local system defined by the *ORIENTATION option referenced.

Second line:

1. Force.
2. Relative displacement.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the spring stiffness as a function of temperature and other predefined field variables.

Data lines to define linear spring behavior for SPRINGA elements with COMPLEX STIFFNESS:

First line:

1. Enter a blank line.

Second line:

1. Real spring stiffness (force per relative displacement).
2. Structural damping factor.
3. Frequency (in cycles per time). Applicable only for *STEADY STATE DYNAMICS, DIRECT; *STEADY STATE DYNAMICS, SUBSPACE PROJECTION; and *STEADY STATE DYNAMICS and *MODAL DYNAMIC analyses that support nondiagonal damping.

Repeat this set of data lines as often as necessary to define the spring stiffness and structural damping factor as a function of frequency.

Data lines to define linear spring behavior for SPRING1 and SPRING2 elements with COMPLEX STIFFNESS:

First line:

1. Degree of freedom with which the springs are associated at their first nodes.
2. For SPRING2 elements give the degree of freedom with which the springs are associated at their second nodes.

If the ORIENTATION parameter is included on the *SPRING option when defining spring elements, the degrees of freedom specified here are in the local system defined by the *ORIENTATION option referenced.

Second line:

1. Real spring stiffness (force per relative displacement).
2. Structural damping factor.
3. Frequency (in cycles per time). Applicable only for *STEADY STATE DYNAMICS, DIRECT; *STEADY STATE DYNAMICS, SUBSPACE PROJECTION; and *STEADY STATE DYNAMICS and *MODAL DYNAMIC analyses that support nondiagonal damping.

Repeat this set of data lines as often as necessary to define the spring stiffness and the structural damping factor as a function of frequency.

18.30 *SRADIATE: Specify surface radiation conditions in heat transfer analysis.

This option is used to apply surface radiation boundary conditions between a nonconcave surface and a nonreflecting environment in fully coupled thermal-stress analysis or to define an approximate cavity radiation interaction. In Abaqus/Standard it is also used for heat transfer and coupled thermal-electrical analyses.

It must be used in conjunction with the *PHYSICAL CONSTANTS option, which is used to define the Stefan-Boltzmann constant.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Interaction module

References:

- "Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual
- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the *AMPLITUDE option that gives the variation of the ambient temperature with time.

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option (see "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted in an Abaqus/Explicit analysis, the reference magnitude is applied immediately at the beginning of the step.

OP

Set OP=MOD (default) for existing *SRADIATE definitions to remain, with this option modifying existing radiation conditions or defining additional radiation conditions.

Set OP=NEW if all existing *SRADIATE definitions applied to the model should be removed.

Data lines to define surface radiation conditions:

First line:

1. Surface name.
2. Radiation type label R.
3. Reference ambient temperature value, θ^0. (Units of θ.)

4. Emissivity, ϵ.

Repeat this data line as often as necessary to define radiation conditions for different surfaces.

Data lines to define approximate cavity radiation (available only in Abaqus/Standard):

First line:

1. Surface name.
2. Radiation type label AVG.
3. Leave this field blank.
4. Emissivity, ϵ.

Repeat this data line as often as necessary to define radiation conditions for different surfaces.

18.31 *STATIC: Static stress/displacement analysis.

This option is used to indicate that the step should be analyzed as a static load step.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Static stress analysis," Section 6.2.2 of the Abaqus Analysis User's Manual
- "Unstable collapse and postbuckling analysis," Section 6.2.4 of the Abaqus Analysis User's Manual
- "Adiabatic analysis," Section 6.5.5 of the Abaqus Analysis User's Manual
- "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual
- "Deformation plasticity," Section 20.2.13 of the Abaqus Analysis User's Manual

No parameters or data lines are used in a linear perturbation analysis.

Optional parameters for a general static analysis:

ADIABATIC

Include this parameter to perform an adiabatic stress analysis. This parameter is relevant only for isotropic metal plasticity materials with a Mises yield surface and when the *INELASTIC HEAT FRACTION option has been specified.

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified via the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual).

CONTINUE

Set CONTINUE=NO (default) to specify that this step will not carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step will carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

DIRECT

This parameter selects direct user control of the incrementation through the step. If this parameter is used, constant increments of the size defined by the first item on the data line are used. If this parameter is omitted, Abaqus/Standard will choose the increments (after trying the user's initial time increment for the first attempt at the first increment).

The parameter can have the value NO STOP. If this value is included, the solution to an increment is accepted after the maximum number of iterations allowed has been completed (as defined by the *CONTROLS option), even if the equilibrium tolerances are not satisfied. Very small increments and a minimum of two iterations are usually necessary if this value is used. *This approach is not recommended; it should be used only in special cases when the analyst has a thorough understanding of how to interpret results obtained in this way.*

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus is not suitable. This parameter must be used in conjunction with the STABILIZE parameter and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

This parameter cannot be used if the RIKS parameter is included.

FULLY PLASTIC

This parameter is relevant only for cases where "fully plastic" analysis is required with deformation theory plasticity. For that purpose set this parameter equal to the name of the element set being monitored for fully plastic behavior.

The step will end when the solutions at all constitutive calculation points in the element set are fully plastic (defined by the equivalent strain being 10 times the offset yield strain). The step will end before this occurs if either the maximum number of increments given on the *STEP option or the time period given on the data line of *STATIC is exceeded.

LONG TERM

Include this parameter to obtain the fully relaxed long-term elastic solution with time-domain viscoelasticity or the long-term elastic-plastic solution for two-layer viscoplasticity. If the LONG TERM parameter is omitted, the instantaneous elastic solution is obtained for time-domain

viscoelasticity and the combined response of the elastic-plastic and elastic-viscous networks is obtained for two-layer viscoplasticity. The parameter is relevant only for time-domain viscoelastic and two-layer viscoplastic materials.

RIKS

Include this parameter to use the modified Riks method for proportional loading cases ("Unstable collapse and postbuckling analysis," Section 6.2.4 of the Abaqus Analysis User's Manual).

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is 2×10^{-4} and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL =0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be overridden by the damping factor.

This parameter cannot be used if the RIKS parameter is included.

Data line for a general static analysis:

First (and only) line:

1. Initial time increment. This value will be modified as required if the automatic time stepping scheme is used or will be used as the constant time increment if the DIRECT parameter is used. If this entry is zero or is not specified, a default value that is equal to the total time period of the step is assumed.
2. Time period of the step. If this entry is zero or is not specified, a default value of 1.0 is assumed.
3. Minimum time increment allowed. Only used for automatic time incrementation. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed.
4. Maximum time increment allowed. Only used for automatic time incrementation. If this value is not specified, no upper limit is imposed.

Data line for the Riks method:

First (and only) line:

1. Initial increment in arc length along the static equilibrium path in scaled load-displacement space, Δl_{in}. If the DIRECT parameter is used, this value will be used as a constant arc length increment. If this entry is zero or is not specified, a default value that is equal to the total arc length of the step is assumed.

2. Total arc length scale factor associated with this step, l_{period}. If this entry is zero or is not specified, a default value of 1.0 is assumed.
3. Minimum arc length increment, Δl_{min}. If this entry is zero, a default value of the smaller of the suggested initial arc length or 10^{-5} times the total arc length is assumed.
4. Maximum arc length increment, Δl_{max}. If this value is not specified, no upper limit is imposed.
5. Maximum value of the load proportionality factor, λ_{end}. This value is used to terminate the step when the load exceeds a certain magnitude.
6. Node number at which the finishing displacement value is being monitored.
7. Degree of freedom being monitored.
8. Value of the total displacement (or rotation) at the node and degree of freedom that, if crossed during an increment, ends the step at the current increment.

18.32 *STEADY STATE CRITERIA: Specify steady-state criteria for terminating a quasi-static uni-directional simulation.

This option is used to specify the norms that must be satisfied to halt a quasi-static uni-directional simulation based on achieving a steady-state condition. It must be used in conjunction with the *STEADY STATE DETECTION option.

Product: Abaqus/Explicit

Type: History data

Level: Step

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- "Steady-state detection," Section 11.9.1 of the Abaqus Analysis User's Manual
- *STEADY STATE DETECTION

There are no parameters associated with this option.

Data lines to define steady-state detection norms SSPEEQ and SSSPRD:

First line:

1. Norm type.
2. Norm tolerance. The default tolerance is .001.
3. Global *X*-coordinate of a point on the exit plane.
4. Global *Y*-coordinate of a point on the exit plane.
5. Global *Z*-coordinate of a point on the exit plane.

Repeat this data line as often as necessary. Each line defines a criterion that must be satisfied to achieve steady state.

Data lines to define steady-state detection norms SSFORC and SSTORQ:

First line:

1. Norm type.
2. Norm tolerance. The default tolerance is .005.
3. Global *X*-coordinate of a point on the exit plane.
4. Global *Y*-coordinate of a point on the exit plane.
5. Global *Z*-coordinate of a point on the exit plane.
6. Rigid body reference node number.

7. First direction cosine of force or torque norm output at the reference node.
8. Second direction cosine of force or torque norm output at the reference node.
9. Third direction cosine of force or torque norm output at the reference node.

Repeat this data line as often as necessary. Each line defines a criterion that must be satisfied to achieve steady state.

18.33 *STEADY STATE DETECTION: Specify steady-state requirements for terminating a quasi-static uni-directional simulation.

This option is used to define the conditions that must be satisfied to determine that steady state has been reached. It must be used in conjunction with the *STEADY STATE CRITERIA option.

Product: Abaqus/Explicit

Type: History data

Level: Step

References:

- "Steady-state detection," Section 11.9.1 of the Abaqus Analysis User's Manual
- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *STEADY STATE CRITERIA

Required parameters:

ELSET

Set this parameter equal to the name of the element set for which this steady-state detection definition is being applied.

SAMPLING

This parameter is used to specify the method used to sample all steady-state norms associated with this option. See "Steady-state detection," Section 11.9.1 of the Abaqus Analysis User's Manual, for more information regarding steady-state norm definitions.

Set SAMPLING=PLANE BY PLANE to calculate the steady-state norms as each plane of elements crosses the exit plane. This method should be used only for non-Eulerian analyses with uniform planes of elements sequentially passing the exit plane.

Set SAMPLING=UNIFORM to calculate the steady-state norms at an interval defined by the time required for material to flow through an element of average length. This interval is determined at the beginning of the step and remains constant throughout the step. This method should be used only for analyses with material flowing in and out of Eulerian boundary regions in the primary direction.

Data line to define primary direction and cutting plane position:

First (and only) line:

1. First direction cosine of primary direction.
2. Second direction cosine of primary direction.

3. Third direction cosine of primary direction.
4. Global *X*-coordinate of a point on the cutting plane.
5. Global *Y*-coordinate of a point on the cutting plane.
6. Global *Z*-coordinate of a point on the cutting plane.

18.34 *STEADY STATE DYNAMICS: Steady-state dynamic response based on harmonic excitation.

This option is used to calculate the system's linearized steady-state response to harmonic excitation.

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Direct-solution steady-state dynamic analysis," Section 6.3.4 of the Abaqus Analysis User's Manual
- "Mode-based steady-state dynamic analysis," Section 6.3.8 of the Abaqus Analysis User's Manual
- "Subspace-based steady-state dynamic analysis," Section 6.3.9 of the Abaqus Analysis User's Manual

Optional and mutually exclusive parameters (used only if the dynamic response is not based on modal superposition):

DIRECT

Include this parameter to compute the steady-state harmonic response directly in terms of the physical degrees of freedom of the model. This usually makes the procedure significantly more expensive, but it can be used if model parameters depend on frequency, if the stiffness of the system is unsymmetric and the unsymmetric terms are important, or if the system contains discrete damping (such as dashpot elements).

SUBSPACE PROJECTION

Include this parameter to compute the steady-state harmonic response on the basis of the subspace projection method. In this case a direct solution is obtained for the model projected onto the eigenvectors obtained in the preceding *FREQUENCY step. This is a cost-effective approach to including consideration of unsymmetric stiffness and frequency-dependent model parameters. It is more expensive than the modal superposition method but less expensive than the direct-solution method.

Set SUBSPACE PROJECTION=ALL FREQUENCIES (default) if the projection of the dynamic equations onto the modal subspace is to be performed at each frequency requested on the data lines.

Set SUBSPACE PROJECTION=CONSTANT if a single projection of the dynamic equations onto the modal subspace is to be used for all frequencies requested on the data lines. The projection is performed using model properties evaluated at the center frequency determined on a logarithmic or linear scale depending on the value of the FREQUENCY SCALE parameter.

Set SUBSPACE PROJECTION=EIGENFREQUENCY if the projections onto the modal subspace of the dynamic equations are to be performed at each eigenfrequency within the requested ranges and at the eigenfrequencies immediately outside these ranges. The projections are then interpolated at each frequency requested on the data lines. The interpolation is done on a logarithmic or linear scale depending on the value of the FREQUENCY SCALE parameter.

Set SUBSPACE PROJECTION=PROPERTY CHANGE to select how often subspace projections onto the modal subspace are performed based on material property changes as a function of frequency. The interpolation is done on a logarithmic or linear scale depending on the value of the FREQUENCY SCALE parameter.

Set SUBSPACE PROJECTION=RANGE if the projections onto the modal subspace of the dynamic equations are to be performed at the lower limit of each frequency range and at the upper limit of the last frequency range. The interpolation is done on a linear scale. This value can be used only with the SIM architecture.

Optional parameters:

DAMPING CHANGE

This parameter is relevant only for SUBSPACE PROJECTION=PROPERTY CHANGE.

Set this parameter equal to the maximum relative change in damping material properties before a new projection is to be performed. The default value is 0.1.

FREQUENCY SCALE

This parameter is relevant only if INTERVAL=EIGENFREQUENCY or INTERVAL=RANGE. If INTERVAL=SPREAD linear scale is used.

Set this parameter equal to LOGARITHMIC (default) or LINEAR to determine whether a logarithmic or linear scale is used for output. If the SUBSPACE PROJECTION parameter is included and is set equal to either EIGENFREQUENCY or PROPERTY CHANGE, the same scale will be used for the interpolation of the subspace projections.

FRICTION DAMPING

This parameter is relevant only if the DIRECT or the SUBSPACE PROJECTION parameter is included.

Set FRICTION DAMPING=NO (default) or YES to ignore or to include friction-induced damping effects at the slipping contact interface for which a velocity differential is imposed.

INTERVAL

Set INTERVAL=EIGENFREQUENCY if the frequency ranges specified on each data line are to be subdivided using the system's eigenfrequencies. This option requires a preceding *FREQUENCY step and is the default if the DIRECT parameter is omitted.

Set INTERVAL=RANGE if the frequency range specified on each data line is to be used directly. This option is the default if the DIRECT parameter is included.

Set INTERVAL=SPREAD to define frequency points around eigenfrequencies found in the frequency ranges specified on each data line. This option requires a preceding *FREQUENCY step.

REAL ONLY

This parameter is relevant only if the DIRECT or the SUBSPACE PROJECTION parameter is included.

Include this parameter if damping terms are to be ignored so that a real, rather than a complex, system matrix is factored. This option can reduce computational time significantly for the DIRECT procedure and, to a lesser extent, for the SUBSPACE PROJECTION procedure.

STIFFNESS CHANGE

This parameter is relevant only for SUBSPACE PROJECTION=PROPERTY CHANGE.

Set this parameter equal to the maximum relative change in stiffness material properties before a new projection is to be performed. The default value is 0.1.

Data lines for a steady-state dynamics analysis if INTERVAL=EIGENFREQUENCY:

First line:

1. Lower limit of frequency range or a single frequency, in cycles/time.
2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency and the remaining data items on the line are ignored.
3. Number of points in the frequency range at which results should be given, including the end points. It is the number of points from the lower limit of the frequency range to the first eigenfrequency in the range; in each interval from eigenfrequency to eigenfrequency; and from the highest eigenfrequency in the range to the high limit of the frequency range. The minimum value is 2. If the value given is less than 2 (or omitted), the default value of 20 points is assumed.
4. Bias parameter. This parameter is useful only if results are requested at four or more frequency points. It is used to bias the results points toward the ends of the intervals so that better resolution is obtained there. This is generally useful since the ends of each interval are the eigenfrequencies where the response amplitudes vary most rapidly. The default bias parameter is 3.0.
5. Frequency scale factor. All the frequency points, except the lower and upper limit of the frequency range, are scaled by this factor. The default frequency scale factor is 1.0.

Repeat this data line as often as necessary to define frequency ranges in which results are required.

Data lines for a steady-state dynamics analysis if INTERVAL=RANGE:

First line:

1. Lower limit of frequency range or a single frequency, in cycles/time.

2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency and the remaining data items on the line are ignored.
3. Total number of points in the frequency range at which results should be given, including the end points. The minimum value is 2. If the value given is less than 2 (or omitted), the default value of 20 points is assumed.
4. Bias parameter. This parameter is useful only if results are requested at four or more frequency points. It is used to bias the results points toward the ends of the intervals so that better resolution is obtained there. The default bias parameter is 1.0 .

Repeat this data line as often as necessary to define frequency ranges in which results are required.

Data lines for a steady-state dynamics analysis if INTERVAL=SPREAD:

First line:

1. Lower limit of frequency range or a single frequency, in cycles/time.
2. Upper limit of frequency range, in cycles/time. If this value is given as zero, it is assumed that results are required at only one frequency and the remaining data items on the line are ignored.
3. Number of equally spaced points around the eigenfrequency at which results should be given, including the eigenfrequency and end points. The minimum value is 3. If the value given is less than 3 (or omitted), the default value of 3 points is assumed.
4. Bias parameter. If the value given is different than 1.0, the bias parameter is reset to 1.0.
5. Frequency scale factor. All the frequency points are scaled by this factor. The default frequency scale factor is 1.0.
6. Frequency spread. Determines the spread as a fractional value of each eigenfrequency in the specified range. The value must be greater than 0.0 and less than 1.0. The default frequency spread is 0.1.

Repeat this data line as often as necessary to define frequency ranges in which results are required.

18.35 *STEADY STATE TRANSPORT: Steady-state transport analysis.

This option is used to indicate that the step should be analyzed as a steady-state transport analysis.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Steady-state transport analysis," Section 6.4.1 of the Abaqus Analysis User's Manual
- "Symmetric model generation," Section 10.4.1 of the Abaqus Analysis User's Manual

Optional parameters:

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified via the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual).

CONTINUE

Set CONTINUE=NO (default) to specify that this step is not to carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step is to carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

DIRECT

This parameter selects direct user control of the incrementation through the step. If this parameter is used, constant increments of the size defined by the first item on the data line are used. If this parameter is omitted, Abaqus/Standard will choose the increments (after trying the user's initial time increment for the first attempt at the first increment).

The parameter can have the value NO STOP. If this value is included, the solution to an increment is accepted after the maximum number of iterations allowed has been completed (as defined by the *CONTROLS option), even if the equilibrium tolerances are not satisfied. Very small increments and a minimum of two iterations are usually necessary if this value is used. *This approach is not recommended; it should be used only in special cases when the analyst has a thorough understanding of how to interpret results obtained in this way.*

ELSET

Set this parameter equal to the name of an element set for which the rigid body motion will be described in a spatial or Eulerian manner. The rest of the elements in the model will be treated in a classical Lagrangian manner. Only one Eulerian element set can be specified in the whole model.

If this parameter is omitted, the rigid body motion in the whole model will be described in a spatial or Eulerian manner.

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus is not suitable. This parameter must be used in conjunction with the STABILIZE parameter and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

INERTIA

Include this parameter to indicate that inertia effects must be accounted for.

Set INERTIA=NO (default) to ignore inertia effects.

Set INERTIA=YES to include inertia effects.

LONG TERM

Include this parameter to indicate that there is no viscoelastic or viscoplastic material response during this step. The solution must be based on the long-term elastic moduli if the material description includes viscoelastic material properties or be based on the long-term response of the elastic-plastic network alone if the two-layer viscoplastic material model is used.

MULLINS

Include this parameter when the Mullins effect material model is used to indicate how the Mullins effect should be applied over the step.

Set MULLINS=RAMP to indicate that the Mullins effect should be ramped up over the time period of the current step.

Set MULLINS=STEP (default) to indicate that the Mullins effect should be applied instantaneously at the beginning of the current step.

PASS BY PASS

Include this parameter to indicate that a quasi-steady-state (pass-by-pass) procedure will be used to obtain the steady-state solution.

If this parameter is omitted, the steady-state solution will be obtained directly.

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is 2×10^{-4} and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL =0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be overridden by the damping factor.

Data line to define a steady-state transport analysis:

First (and only) line:

1. Initial time increment. This value will be modified as required if the automatic time stepping scheme is used or will be used as the constant time increment if the DIRECT parameter is used.
2. Time period of the step.
3. Minimum time increment allowed. Only used for automatic time incrementation. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed.
4. Maximum time increment allowed. Only used for automatic time incrementation. If this value is not specified, no upper limit is imposed.

18.36 *STEP: Begin a step.

This option is used to begin each step definition. It must be followed by a procedure definition option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Model

Abaqus/CAE: Step module

Beginning a step in an Abaqus/Standard analysis

References:

- "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual
- "Convergence criteria for nonlinear problems," Section 7.2.3 of the Abaqus Analysis User's Manual
- "Design sensitivity analysis," Section 16.1.1 of the Abaqus Analysis User's Manual
- *END STEP

Optional parameters:

AMPLITUDE

This parameter defines the default amplitude variation for loading magnitudes during the step.

Set AMPLITUDE=STEP if the load is to be applied instantaneously at the start of the step and remain constant throughout the step.

Set AMPLITUDE=RAMP if the load magnitude is to vary linearly over the step, from the value at the end of the previous step (or zero, at the start of the analysis) to the value given on the loading option.

If this parameter is omitted, the default amplitude choice depends on the procedure chosen, as shown in "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual. The default amplitude variation can be overwritten for individual loadings by using the AMPLITUDE parameter on the loading options ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual).

This parameter is rarely needed, and changing the defaults may cause problems. For example, the automatic load incrementation scheme in procedures without a real time scale (such as the *STATIC option) applies the loads gradually by incrementing the normalized time scale. The use of AMPLITUDE=STEP specifies that the entire load will be applied immediately, so Abaqus/Standard may not be able to choose suitable small increments if the loading causes strongly nonlinear response.

CONVERT SDI

This parameter determines how severe discontinuities (such as contact changes) are accounted for during nonlinear analysis.

Set CONVERT SDI=YES (default) to use local convergence criteria to determine whether a new iteration is needed. Abaqus/Standard will determine the maximum penetration and estimated force errors associated with severe discontinuities and check whether these errors are within the tolerances. Hence, a solution may converge if the severe discontinuities are small.

Set CONVERT SDI=NO to force a new iteration if severe discontinuities occur during an iteration, regardless of the magnitude of the penetration and force errors. This option also changes some time incrementation parameters and uses different criteria to determine whether to do another iteration or to make a new attempt with a smaller increment size.

If the CONVERT SDI parameter is omitted, Abaqus/Standard will use the value specified in the previous general analysis step. An exception is the first new step of a restart analysis, which will use CONVERT SDI=YES by default regardless of the setting in the previous step.

This parameter has no relevance and will be ignored for heat transfer analysis and linear perturbation steps.

DSA

This parameter applies only to Abaqus/Design.

Set DSA=YES to activate design sensitivity analysis for the step. Once DSA is activated in a general step, it remains active in all subsequent general steps until it is deactivated in a subsequent general step by setting DSA=NO. Once DSA is activated in a perturbation step, it remains active in all subsequent consecutive perturbation steps until it is deactivated in a subsequent consecutive perturbation step. However, if DSA is activated in a step whose procedure is not supported for DSA, DSA will be deactivated until it is activated again by setting DSA=YES.

EXTRAPOLATION

This parameter is useful only for nonlinear analyses.

Set EXTRAPOLATION=LINEAR (default for procedures other than *DYNAMIC, APPLICATION=TRANSIENT FIDELITY) to indicate that the process is essentially monotonic, so that Abaqus/Standard should use a 100% linear extrapolation, in time, of the previous incremental solution to begin the nonlinear equation solution for the current increment (a 1% extrapolation is used with the Riks method).

Set EXTRAPOLATION=PARABOLIC to indicate that the process should use a quadratic displacement-based extrapolation, in time, of the previous two incremental solutions to begin the nonlinear equation solution for the current increment.

Set EXTRAPOLATION=VELOCITY PARABOLIC (available for *DYNAMIC procedure only and default for the *DYNAMIC APPLICATION=TRANSIENT FIDELITY procedure) to indicate that the process should use a quadratic velocity-based extrapolation, in time, of the previous incremental solutions to begin the nonlinear equation solution for the current increment.

Set EXTRAPOLATION=NO to suppress any extrapolation.

INC

Set this parameter equal to the maximum number of increments in a step (or in a single loading cycle for direct cyclic analysis). This value is only an upper bound. The default value is 100.

The INC parameter has no effect in procedures where automatic incrementation cannot be used (for example, *BUCKLE, *STEADY STATE DYNAMICS, and *MODAL DYNAMIC).

NAME

Set this parameter equal to a label that will be used to refer to the step on the output database. Step names in the same input file must be unique. Step names from the original input file can be reused in a restart input file.

NLGEOM

Omit this parameter or set NLGEOM=NO to perform a geometrically linear analysis during the current step. Include this parameter or set NLGEOM=YES to indicate that geometric nonlinearity should be accounted for during the step (stress analysis and fully coupled thermal-stress analysis only). Once the NLGEOM option has been switched on, it will be active during all subsequent steps in the analysis.

PERTURBATION

Include this parameter to indicate that this is a linear perturbation step. For this type of analysis Abaqus/Standard expects that load, boundary, and temperature changes should be given and that the results will be changes relative to the previous step. *Please read the discussions in "General and linear perturbation procedures," Section 6.1.2 of the Abaqus Analysis User's Manual, "Mesh-to-mesh solution mapping," Section 12.4.1 of the Abaqus Analysis User's Manual, and "Applying loads: overview," Section 30.4.1 of the Abaqus Analysis User's Manual, before using this option.*

SOLVER

Set SOLVER=ITERATIVE to use the iterative linear equation solver. *Please read the discussion in "Iterative linear equation solver," Section 6.1.5 of the Abaqus Analysis User's Manual, before using this option.*

If this parameter is omitted, the default direct sparse solver is used.

UNSYMM

Set UNSYMM=YES to indicate that unsymmetric matrix storage and solution should be used.

Set UNSYMM=NO to indicate that symmetric storage and solution should be used.

The default value for this parameter depends on the model and procedure options used. The user is allowed to change the default value only in certain cases. If the UNSYMM parameter is not used in such cases, Abaqus/Standard will use the value specified in the previous general analysis step. See "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual, for a more detailed discussion of the use of this parameter.

Optional data lines:

First line:

1. Subheading for this step.

The subheading can be several lines long, but only the first 80 characters of the first line will be saved and printed as a subheading.

Beginning a step in an Abaqus/Explicit analysis

References:

- "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual
- *END STEP

Optional parameters:

NAME

Set this parameter equal to the name used to identify the step on the output database. Step names in the same input file must be unique.

NLGEOM

Set NLGEOM=YES (default) to indicate that geometric nonlinearity should be accounted for during the step (stress analysis and fully coupled thermal-stress analysis only). Once the NLGEOM option has been switched on, it will be active during all subsequent steps in the analysis. Set NLGEOM=NO to perform a geometrically linear analysis during the current step.

The default value for the NLGEOM parameter in an Abaqus/Explicit analysis is YES unless the Abaqus/Explicit analysis is an import analysis, in which case the default value of the NLGEOM parameter is the same as the value of the parameter at the time of import.

Optional data lines:

First line:

1. Subheading for this step.

The subheading can be several lines long, but only the first 80 characters of the first line will be saved and printed as a subheading.

18.37 *SUBCYCLING: Define a subcycling zone.

This option is used to define a subcycling zone.

Product: Abaqus/Explicit

Type: Model data

Level: Model

References:

- “Explicit dynamic analysis,” Section 6.3.3 of the Abaqus Analysis User’s Manual
- “Fully coupled thermal-stress analysis,” Section 6.5.4 of the Abaqus Analysis User’s Manual
- “Selective subcycling,” Section 11.8.1 of the Abaqus Analysis User’s Manual

Required parameter:

ELSET

Set this parameter equal to the name of the element set to define a subcycling zone.

There are no data lines associated with this option.

18.38 *SUBMODEL: Specify driven boundary nodes in submodeling analysis.

This option is used to specify the total list of "driven regions" for a submodel.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Load module and model attribute

Reference:

- "Submodeling: overview," Section 10.2.1 of the Abaqus Analysis User's Manual

Optional, mutually exclusive parameters:

ACOUSTIC TO STRUCTURE

Include this parameter if the submodel will be driven at the specified surface by the acoustic pressure from a global coupled acoustic-structural model.

SHELL TO SOLID

Include this parameter if a solid element submodel will be driven by a global shell model. If this parameter is included, all driven nodes must be on solid elements and must be located in regions modeled with shell elements in the global model. If this parameter is included on any *SUBMODEL option in an input file, it must be included on all *SUBMODEL options in the input file.

Required parameter for SHELL TO SOLID submodeling:

SHELL THICKNESS

If the OFFSET parameter is not used on the *SHELL SECTION or *SHELL GENERAL SECTION option in the global model, set this parameter equal to the maximum value of the shell thickness in the global model (given in the units used in the model). If the OFFSET parameter is used in the global model, set this parameter equal to twice the maximum distance from the reference surface to the top or bottom shell surface.

Optional parameters:

ABSOLUTE EXTERIOR TOLERANCE

Set this parameter equal to the absolute value (given in the units used in the model) by which a driven node of the submodel may lie outside the region of the elements of the global model. If this parameter is not used or has a value of 0.0, the EXTERIOR TOLERANCE will apply. For

shell-to-solid submodeling the driven node may lie within a region defined by half the value of the SHELL THICKNESS parameter plus the exterior tolerance.

EXTERIOR TOLERANCE

Set this parameter equal to the fraction of the average element size in the global model by which a driven node of the submodel may lie outside the region of the elements of the global model. The default is 0.05. For shell-to-solid submodeling the driven node may lie within a region defined by half the value of the SHELL THICKNESS parameter plus the exterior tolerance.

If both tolerance parameters are specified by the user, Abaqus uses the tighter tolerance.

GLOBAL ELSET

Set this parameter equal to the name of the element set in the global model that will be searched for elements whose responses will be used to drive the submodel. If this parameter is omitted, Abaqus will search all elements in the global model that lie in the vicinity of the submodel.

This parameter must be used with the ACOUSTIC TO STRUCTURE parameter when the acoustic pressures act on both sides of a shell.

INTERSECTION ONLY

Include this parameter to specify that Abaqus ignore driven nodes found to lie outside the region of elements of the global model, after accounting for the exterior search tolerance.

This parameter can be used only with TYPE=NODE. This parameter cannot be used with the ACOUSTIC TO STRUCTURE or SHELL TO SOLID parameters.

TYPE

This parameter applies only to Abaqus/Standard analyses. It determines whether global model communication to the submodel occurs through nodes or through surfaces.

Set TYPE=NODE (default) for node-based submodeling. Node-based submodel definitions will be accompanied by *BOUNDARY, SUBMODEL definitions.

Set TYPE=SURFACE for surface-based submodeling. This parameter setting cannot be used with the ACOUSTIC TO STRUCTURE, SHELL TO SOLID, or SHELL THICKNESS parameters. Surface-based submodel definitions will be accompanied by *DSLOAD, SUBMODEL definitions.

Data lines to define the driven boundary for general and shell-to-solid submodeling:

First line:

1. List of nodes or node set labels. Up to 16 entries are allowed per line.

Repeat this data line as often as necessary.

Data lines to define the driven boundary for acoustic-to-structure submodeling:

First line:

1. The name of the element-based structural surface.

Repeat this data line as often as necessary.

Data lines to define the driven surfaces for surface-based submodeling:

First line:

1. The name of the element-based surface.

Repeat this data line as often as necessary.

18.39 *SUBSTRUCTURE COPY: Copy a substructure definition.

This option is used to copy a substructure definition from one library to another or from one substructure identifier to another within one library.

Product: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Reference:

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Manual

Required parameters:

NEW TYPE

Set this parameter equal to the TYPE identifier of the substructure being created.

OLD TYPE

Set this parameter equal to the TYPE identifier of the substructure being copied.

Optional parameters:

NEW LIBRARY

Set this parameter equal to the name of the substructure library in which the substructure is stored. The default library name is *jobname*.

OLD LIBRARY

Set this parameter equal to the name of the substructure library from which the substructure is being copied. The default library name is *jobname*.

There are no data lines associated with this option.

18.40 *SUBSTRUCTURE DELETE: Remove a substructure from the substructure library.

This option is used to delete a substructure from a substructure library.

Product: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Reference:

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Manual

Required parameter:

TYPE

Set this parameter equal to the TYPE identifier of the substructure being removed.

Optional parameter:

LIBRARY

Set this parameter equal to the name of the substructure library from which the substructure is being deleted. The default library name is *jobname*.

There are no data lines associated with this option.

18.41 *SUBSTRUCTURE DIRECTORY: List information about the substructures on a substructure library.

This option is used to provide a summary of information about the substructures stored on a substructure library.

Product: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Reference:

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Manual

Required parameter:

LIBRARY

Set this parameter equal to the name of the substructure library.

There are no data lines associated with this option.

18.42 *SUBSTRUCTURE GENERATE: Substructure generation analysis.

This option is used to indicate that the step should be analyzed as a substructure generation step.

Product: Abaqus/Standard

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Reference:

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Manual

Required parameter:

TYPE

Set this parameter equal to the identifier to be assigned to this substructure in a substructure library. The identifier must be *Z* followed by a number that cannot exceed 9999.

Substructure identifiers must be unique within a library. If a substructure already exists in the library with this same identifier, the analysis will terminate with an error message unless the OVERWRITE parameter is specified.

Optional parameters:

ELSET

If element output recovery is needed, including all element nodes in the selective recovery node set generally is insufficient since an element can have Abaqus internal nodes.

Set this parameter equal to the name of the element set that contains all the elements in the regions of the substructure where you want to recover results.

GRAVITY LOAD

Set GRAVITY LOAD=YES to calculate the substructure's gravity load vectors. The default is GRAVITY LOAD=NO.

LIBRARY

Set this parameter equal to the name of the substructure library on which the substructure data will be written. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such library names. The default library name is *jobname*.

MASS MATRIX

Set MASS MATRIX=YES to calculate the substructure's reduced mass matrix. The default is MASS MATRIX=NO.

NSET

Set this parameter equal to the name of the node set that contains the nodes of the substructure where you want to recover results. This node set must contain all nodes for which node output can be requested in a substructure usage analysis.

If the NSET parameter is omitted but the ELSET parameter is used, the recovery matrix corresponding to all the element nodes in the specified element set is generated. If both the NSET and ELSET parameters are used, the recovery matrix for the union of the node set and the set of all the element nodes for all the elements in the element set is generated. If both the NSET and ELSET parameters are omitted, the recovery matrix for all eliminated nodes is generated (default case).

OVERWRITE

Include this parameter to overwrite an existing substructure with the same TYPE identifier in the library. The default is no overwrite.

PROPERTY EVALUATION

Set this parameter equal to the frequency at which to evaluate frequency-dependent properties for viscoelasticity, springs, and dashpots during the substructure generation. If this parameter is omitted, Abaqus/Standard will evaluate the stiffness associated with frequency-dependent springs and dashpots at zero frequency and will not consider the stiffness contributions from frequency-domain viscoelasticity in the *SUBSTRUCTURE GENERATE step.

RECOVERY MATRIX

Set RECOVERY MATRIX=NO to specify that output of element or nodal information is not available within this substructure. The default is RECOVERY MATRIX=YES, indicating that recovery of eliminated variables is possible for most analysis procedures.

If RECOVERY MATRIX=NO, the NSET and ELSET parameters are ignored.

STRUCTURAL DAMPING MATRIX

Set STRUCTURAL DAMPING MATRIX=YES to calculate the substructure's reduced structural damping matrix. The default is STRUCTURAL DAMPING MATRIX=NO.

VISCOUS DAMPING MATRIX

Set VISCOUS DAMPING MATRIX=YES to calculate the substructure's reduced viscous damping matrix. The default is VISCOUS DAMPING MATRIX=NO.

There are no data lines associated with this option.

18.43 *SUBSTRUCTURE LOAD CASE: Begin the definition of a substructure load case.

This option is used to begin the definition of a substructure load case for the substructure currently being generated. It can be used only in a *SUBSTRUCTURE GENERATE analysis.

Product: Abaqus/Standard

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Manual
- *SLOAD

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the load case in *SLOAD option specifications when applying loads to the substructure during an analysis.

To define the loads:

Enter any mechanical loading options ("Concentrated loads," Section 30.4.2 of the Abaqus Analysis User's Manual, and "Distributed loads," Section 30.4.3 of the Abaqus Analysis User's Manual) or thermal loading options ("Thermal loads," Section 30.4.4 of the Abaqus Analysis User's Manual) to define the loads forming the load case. Specify a magnitude for each load. This magnitude will be scaled by a magnitude and amplitude reference specified in the *SLOAD option. The load case definition continues until an option is encountered that is not one of the loading options. If boundary conditions are included in a *SUBSTRUCTURE LOAD CASE, they are always active, even if the *SLOAD option is not used.

18.44 *SUBSTRUCTURE MATRIX OUTPUT: Write a substructure's recovery matrix, reduced stiffness matrix, mass matrix, load case vectors, and gravity load vectors to a file.

This option is used to write a substructure's recovery matrix, reduced stiffness matrix, mass matrix, load case vectors, and gravity load vectors to a file. It can be used only in a *SUBSTRUCTURE GENERATE analysis.

Product: Abaqus/Standard

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Reference:

- "Defining substructures," Section 10.1.2 of the Abaqus Analysis User's Manual

Optional parameters:

FILE NAME

This parameter is intended for use with OUTPUT FILE=USER DEFINED.

Set this parameter equal to the name of the file (without an extension) to which the data will be written. The extension **.mtx** will be added to the file name provided by the user; see "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names.

If the OUTPUT FILE parameter is omitted or set equal to RESULTS FILE, use of the FILE NAME parameter overrides the OUTPUT FILE setting; the data will be written to the named file, not to the results (**.fil**) file. If OUTPUT FILE=ODB, the FILE NAME parameter is ignored and the data will be written to the output database (**.odb**) file.

GRAVITY LOAD

Set GRAVITY LOAD=YES to write the substructure gravity load vectors (only available if the gravity load vectors are requested with the *SUBSTRUCTURE GENERATE option). The default is GRAVITY LOAD=NO.

MASS

Set MASS=YES to write the substructure mass matrix (only available if the mass matrix is requested with the *SUBSTRUCTURE GENERATE option). The default is MASS=NO.

OUTPUT FILE

Set OUTPUT FILE=RESULTS FILE (default) to write the data to the results (**.fil**) file in the format specified in "Results file output format," Section 5.1.2 of the Abaqus Analysis User's Manual.

Set OUTPUT FILE=USER DEFINED to write the results to a user-specified file in the format of the *USER ELEMENT, LINEAR option ("User-defined elements," Section 29.16.1 of the Abaqus Analysis User's Manual). The name of the file is specified using the FILE NAME parameter.

Set OUTPUT FILE=ODB to write the data to the output database (**.odb**) file.

RECOVERY MATRIX

Set RECOVERY MATRIX=YES to write the substructure recovery matrix (only available if the recovery matrix is requested with the *SUBSTRUCTURE GENERATE option). The default is RECOVERY MATRIX=NO.

SLOAD

Set SLOAD=YES to write the substructure load case vectors. The default is SLOAD=NO.

STIFFNESS

Set STIFFNESS=YES to write the substructure stiffness matrix. The default is STIFFNESS=NO.

There are no data lines associated with this option.

18.45 *SUBSTRUCTURE PATH: Enter into a substructure to obtain output or return back from a previously entered substructure.

This option is used to navigate through "levels" of substructures to obtain output of results.

Product: Abaqus/Standard

Type: History data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Reference:

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Manual

At least one of the following parameters is required:

ENTER ELEMENT

Include this parameter if output requests within a substructure are to follow. Set this parameter equal to the element number used for the substructure at the usage level at which Abaqus/Standard is operating before the appearance of this option.

LEAVE

Include this parameter to leave a substructure and go back up to its usage level.

There are no data lines associated with this option.

18.46 *SUBSTRUCTURE PROPERTY: Translate, rotate, and/or reflect substructures.

This option is used to define properties for a substructure. It is required for all substructures in a model.

Product: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Reference:

- "Using substructures," Section 10.1.1 of the Abaqus Analysis User's Manual

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the substructures for which properties are being defined.

Optional parameter:

POSITION TOL

Set this parameter equal to the tolerance on the distance between usage level nodes and the corresponding substructure nodes. If this parameter is omitted, the default is a tolerance of 10^{-4} times the largest overall dimension within the substructure. If the parameter is given with a value of 0.0, the position of the retained nodes is not checked.

Data line to translate a substructure:

First (and only) line:

1. Value of the translation to be applied in the global X-direction.
2. Value of the translation to be applied in the global Y-direction.
3. Value of the translation to be applied in the global Z-direction.

Data lines to translate and/or rotate a substructure:

First line:

1. Value of the translation to be applied in the global X-direction.
2. Value of the translation to be applied in the global Y-direction.
3. Value of the translation to be applied in the global Z-direction.

Enter values of zero to apply a pure rotation.

Second line:

1. Global *X*-coordinate of point *a* on the axis of rotation (see Figure 18.46–1).
2. Global *Y*-coordinate of point *a* on the axis of rotation.
3. Global *Z*-coordinate of point *a* on the axis of rotation.
4. Global *X*-coordinate of point *b* on the axis of rotation.
5. Global *Y*-coordinate of point *b* on the axis of rotation.
6. Global *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

Data lines to translate and/or reflect a substructure:

First line:

1. Value of the translation to be applied in the global *X*-direction.
2. Value of the translation to be applied in the global *Y*-direction.
3. Value of the translation to be applied in the global *Z*-direction.

Enter values of zero to apply a pure reflection.

Second line:

1. Enter a blank line.

Third line:

1. Global *X*-coordinate of point *a* in the plane of reflection (see Figure 18.46–2).
2. Global *Y*-coordinate of point *a* in the plane of reflection.
3. Global *Z*-coordinate of point *a* in the plane of reflection.
4. Global *X*-coordinate of point *b* in the plane of reflection.
5. Global *Y*-coordinate of point *b* in the plane of reflection.
6. Global *Z*-coordinate of point *b* in the plane of reflection.

Fourth line:

1. Global *X*-coordinate of point *c* in the plane of reflection.
2. Global *Y*-coordinate of point *c* in the plane of reflection.
3. Global *Z*-coordinate of point *c* in the plane of reflection.

Data lines to translate, rotate, and reflect a substructure:

First line:

1. Value of the translation to be applied in the global *X*-direction.
2. Value of the translation to be applied in the global *Y*-direction.
3. Value of the translation to be applied in the global *Z*-direction.

Second line:

1. Global *X*-coordinate of point *a* on the axis of rotation (see Figure 18.46–1).
2. Global *Y*-coordinate of point *a* on the axis of rotation.
3. Global *Z*-coordinate of point *a* on the axis of rotation.
4. Global *X*-coordinate of point *b* on the axis of rotation.
5. Global *Y*-coordinate of point *b* on the axis of rotation.
6. Global *Z*-coordinate of point *b* on the axis of rotation.
7. Angle of rotation about the axis *a*–*b*, in degrees.

Third line:

1. Global *X*-coordinate of point *a* in the plane of reflection (see Figure 18.46–2).
2. Global *Y*-coordinate of point *a* in the plane of reflection.
3. Global *Z*-coordinate of point *a* in the plane of reflection.
4. Global *X*-coordinate of point *b* in the plane of reflection.
5. Global *Y*-coordinate of point *b* in the plane of reflection.
6. Global *Z*-coordinate of point *b* in the plane of reflection.

Fourth line:

1. Global *X*-coordinate of point *c* in the plane of reflection.
2. Global *Y*-coordinate of point *c* in the plane of reflection.
3. Global *Z*-coordinate of point *c* in the plane of reflection.

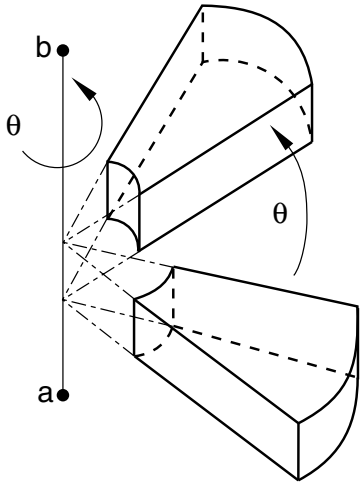


Figure 18.46–1 Substructure rotation.

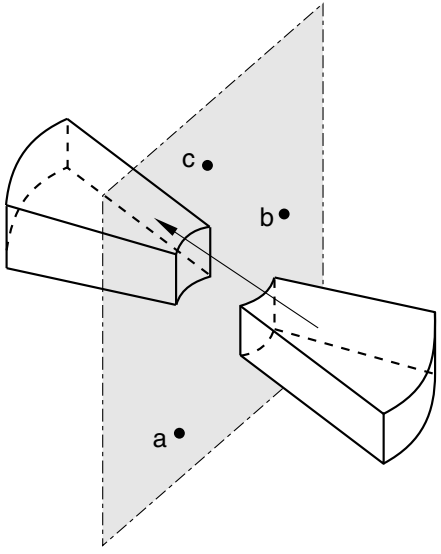


Figure 18.46–2 Substructure reflection. Points *a*, *b*, and *c* cannot be colinear.

18.47 *SURFACE: Define a surface or region in a model.

This option is used to define surfaces for contact simulations, tie constraints, fasteners, and coupling, as well as regions for distributed surface loads, acoustic radiation, acoustic impedance, and output of integrated quantities on a surface. In Abaqus/Standard it is also used to define surfaces for cavity radiation analysis and assembly loads. In Abaqus/Explicit this option can also be used to define boundary regions on adaptive mesh domains.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Element-based surfaces are supported by the Surface toolset. Node-based surfaces are not supported; if node-based surfaces are imported into Abaqus/CAE, they are treated as sets.

References:

- "Surfaces: overview," Section 2.3.1 of the Abaqus Analysis User's Manual
- "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Manual
- "Node-based surface definition," Section 2.3.3 of the Abaqus Analysis User's Manual
- "Analytical rigid surface definition," Section 2.3.4 of the Abaqus Analysis User's Manual
- "Eulerian surface definition," Section 2.3.5 of the Abaqus Analysis User's Manual
- "Operating on surfaces," Section 2.3.6 of the Abaqus Analysis User's Manual
- "Eulerian analysis," Section 13.1.1 of the Abaqus Analysis User's Manual
- "Contact interaction analysis: overview," Section 32.1.1 of the Abaqus Analysis User's Manual
- "RSURFU," Section 1.1.16 of the Abaqus User Subroutines Reference Manual

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the surface.

Required parameter for cavity radiation simulations:

PROPERTY

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of the *SURFACE PROPERTY definition associated with this surface. See "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual.

Optional parameters:

COMBINE

Set COMBINE=UNION to create a surface based on the union of two or more surfaces of the same type.

Set COMBINE=INTERSECTION to create a surface based on the intersection of two surfaces of the same type.

Set COMBINE=DIFFERENCE to create a surface based on the difference of two surfaces of the same type (the second surface is subtracted from the first).

Only the NAME parameter and, in cavity radiation simulations, the PROPERTY parameter can be used in conjunction with this parameter.

CROP

Include this parameter to create a new surface that will contain only those faces from an existing surface that have nodes in a specified rectangular box.

Only the NAME parameter and, in cavity radiation simulations, the PROPERTY parameter can be used in conjunction with this parameter.

DEFINITION

This parameter is relevant only for surfaces defined using TYPE=CUTTING SURFACE.

Set DEFINITION=COORDINATES (default) to define the cutting plane by giving the coordinates of a point on the cutting plane and the normal to the cutting plane.

Set DEFINITION=NODES to define the cutting plane by giving global node numbers for point a on the cutting plane and point b that lies off the cutting plane with the cutting plane normal determined by the vector from a to b.

FILLET RADIUS

This parameter can be used with TYPE=SEGMENTS, TYPE=CYLINDER, or TYPE=REVOLUTION to define a radius of curvature to smooth discontinuities between adjoining straight-line segments, adjoining circular-arc segments, and adjoining straight-line and circular-arc segments.

INTERNAL

Abaqus/CAE uses the INTERNAL parameter to identify surfaces that are created internally. The INTERNAL parameter is used only in models defined in terms of an assembly of part instances. The default is to omit the INTERNAL parameter.

REGION TYPE

This parameter is relevant only for surfaces defined on the boundary of an adaptive mesh domain. A surface defined in the interior of an adaptive mesh domain will move independently of the material unless the surface is constrained by mesh constraints. See “Defining ALE adaptive mesh domains in Abaqus/Explicit,” Section 12.2.2 of the Abaqus Analysis User’s Manual.

Set REGION TYPE=LAGRANGIAN to create a Lagrangian boundary region. The edge of a Lagrangian boundary region will follow the material while allowing adaptive meshing along the edge and within the interior of the region.

Set REGION TYPE=SLIDING (default) to create a sliding boundary region. The edge of a sliding boundary region will slide over the material. Adaptive meshing will occur on the edge and within the interior of the region. Mesh constraints are typically applied on the edge of a sliding boundary region to fix it spatially.

Set REGION TYPE=EULERIAN to create an Eulerian boundary region in an adaptive mesh domain. This option is used to create a boundary region across which material can flow. Mesh constraints must be used normal to an Eulerian boundary region to allow material to flow through the region. If no mesh constraints are applied, an Eulerian boundary region will behave in the same way as a sliding boundary region.

TRIM

Set TRIM=YES to invoke trimming of open free surfaces. Set TRIM=NO to suppress surface trimming. The default value is TRIM=YES unless the surface is used as a master surface in a finite-sliding contact formulation in Abaqus/Standard or the surface is used with the contact pair algorithm in Abaqus/Explicit. TRIM=YES has no effect on surfaces used with the contact pair algorithm in Abaqus/Explicit.

TYPE

Set TYPE=ELEMENT (default) to define a free surface automatically for the elements specified or to define a surface on the elements by using element face identifiers.

Set TYPE=NODE to define a surface by specifying a list of nodes or node set labels.

Set TYPE=SEGMENTS to create a two-dimensional analytical surface in the (X, Y) plane for planar models or in the (r, z) plane for axisymmetric models by defining connected line segments.

Set TYPE=CYLINDER to define a three-dimensional analytical surface by sweeping connected line segments defined in a local (x, y) plane along a specified generator vector.

Set TYPE=REVOLUTION to define a three-dimensional analytical surface by providing connected line segments, which are given in an (r, z) plane and are rotated about an axis.

Set TYPE=CUTTING SURFACE to generate an interior element-based surface using a cutting plane passing through an element set. The generated surface is an approximation to the cutting plane.

Set TYPE=EULERIAN MATERIAL to define a surface on the exterior boundary of an Eulerian material instance. This option applies only to Abaqus/Explicit.

Set TYPE=USER to define an analytical surface via user subroutine **RSURFU** in Abaqus/Standard.

Additional optional parameters used for contact pair analyses in Abaqus/Explicit:

MAX RATIO

Set this parameter equal to the upper bound of the ratio of a facet's contact thickness to its minimum edge (or diagonal) length. This ratio is computed after the SCALE THICK parameter is applied to the contact thickness. The contact thickness for individual facets will be adjusted, if necessary, to conform to this maximum ratio.

If this parameter is omitted, no upper bound will be enforced. If this parameter is included without a value specified, the default value is 0.6.

NO OFFSET

Include this parameter to indicate that this surface will ignore the midplane offset of any shell, membrane, or rigid elements that form the surface.

NO THICK

Include this parameter to indicate that this surface will ignore the thickness of any shell, membrane, or rigid elements that form the surface. A surface defined with this parameter cannot be used to define a double-sided surface or for self-contact.

SCALE THICK

This parameter applies only when the surface is used with the *CONTACT PAIR option.

Set this parameter equal to the amount by which to scale the thicknesses of the underlying elements to compute the contact thicknesses. The default is 1.0 (i.e., no scaling).

Data lines for COMBINE=UNION:

First line:

1. List of surfaces.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data line for COMBINE=INTERSECTION or COMBINE=DIFFERENCE:

First (and only) line:

1. First surface name.
2. Second surface name.

For COMBINE=DIFFERENCE the second surface is subtracted from the first.

Data lines to define a surface when the CROP parameter is included:

First line:

1. Surface name.

Second line:

1. *X*-coordinate of the lower box corner.
2. *Y*-coordinate of the lower box corner.
3. *Z*-coordinate of the lower box corner.
4. *X*-coordinate of the opposite box corner.
5. *Y*-coordinate of the opposite box corner.
6. *Z*-coordinate of the opposite box corner.

Third line (optional):

1. *X*-coordinate of the first point defining the orientation.
2. *Y*-coordinate of the first point defining the orientation.

3. *Z*-coordinate of the first point defining the orientation.
4. *X*-coordinate of the second point defining the orientation.
5. *Y*-coordinate of the second point defining the orientation.
6. *Z*-coordinate of the second point defining the orientation.

Data lines to define a surface using elements or element sets when the TYPE=ELEMENT parameter is used:

First line:

1. Element set name or element number. In Abaqus/Explicit a blank data line can be specified to automatically generate the exterior (free) faces of every element in the model.
2. Face or edge identifier label (see "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Manual, for the face and edge identifiers for various elements) or the "word" EDGE (optional).

Repeat this data line as often as necessary to define the surface.

Data lines to define a surface using nodes or node sets when the TYPE=NODE parameter is used:

First line:

1. Node set name or node number.
2. Cross-sectional area or distributing weight factor. In Abaqus/Standard contact calculations, the default is the area specified in the associated *SURFACE INTERACTION option if the surface is defined in a contact pair; otherwise, a unit area is used. In Abaqus/Explicit the cross-sectional area used for contact pair calculations for node-based surface nodes is always set to 1.0 regardless of the value specified here. If the surface is used in a *COUPLING or *SHELL TO SOLID COUPLING definition, the default distributing weight factor is zero in Abaqus/Standard and 1.0 in Abaqus/Explicit.

Repeat this data line as often as necessary to define the surface.

Data lines to define a surface using a plane cutting through the given element sets when TYPE=CUTTING SURFACE, DEFINITION=COORDINATES:

First line:

1. *X*-coordinate of a point on the cutting plane in the initial configuration.
2. *Y*-coordinate of a point on the cutting plane in the initial configuration.
3. *Z*-coordinate of a point on the cutting plane in the initial configuration.
4. *X*-component of a normal to the cutting plane in the initial configuration.
5. *Y*-component of a normal to the cutting plane in the initial configuration.
6. *Z*-component of a normal to the cutting plane in the initial configuration.

Second line:

1. List of elements or element set labels to be cut by the cutting plane to generate an element-based surface that is an approximation to the cutting plane. A blank data line can be specified to generate a surface by cutting the whole model.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data lines to define a surface using a plane cutting through the given element sets when TYPE=CUTTING SURFACE, DEFINITION=NODES:

First line:

1. Node number of the node at point *a*.
2. Node number of the node at point *b*.

Second line:

1. List of elements or element set labels to be cut by the cutting plane to generate an element-based surface that is an approximation to the cutting plane. A blank data line can be specified to generate a surface by cutting the whole model.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

Data line to define surfaces created with TYPE=EULERIAN MATERIAL:

First line:

1. Name of the material instance as defined in the *EULERIAN SECTION. Abaqus/Explicit will automatically create a surface on the exterior of the given material.

No data lines are needed for TYPE=USER.

Data lines to define surfaces created with TYPE=SEGMENTS:

First line:

1. The "word" START.
2. Global *X*-coordinate or *r*-coordinate of the starting point of the line segments.
3. Global *Y*-coordinate or *z*-coordinate of the starting point of the line segments.

Second and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the analytical surface.

Data lines to define surfaces created with TYPE=CYLINDER:

First line (leave blank if this surface is being defined within a part):

1. Global *X*-coordinate of point *a*, the origin of the local (x, y) system (see Figure 18.47–1).
2. Global *Y*-coordinate of point *a*, the origin of the local (x, y) system.

3. Global *Z*-coordinate of point *a*, the origin of the local (x, y) system.
4. Global *X*-coordinate of point *b* on the local *x*-axis.
5. Global *Y*-coordinate of point *b* on the local *x*-axis.
6. Global *Z*-coordinate of point *b* on the local *x*-axis.

Second line (leave blank if this surface is being defined within a part):

1. Global *X*-coordinate of point *c* on the local cylinder generator vector.
2. Global *Y*-coordinate of point *c* on the local cylinder generator vector.
3. Global *Z*-coordinate of point *c* on the local cylinder generator vector.

Third line:

1. The "word" START.
2. Local *x*-coordinate of the starting point of the line segments.
3. Local *y*-coordinate of the starting point of the line segments.

Fourth and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the analytical surface.

Data lines to define surfaces created with TYPE=REVOLUTION:

First line (leave blank if this surface is being defined within a part):

1. Global *X*-coordinate of point *a*, the origin of the local (r, z) system (see Figure 18.47–2).
2. Global *Y*-coordinate of point *a*, the origin of the local (r, z) system.
3. Global *Z*-coordinate of point *a*, the origin of the local (r, z) system.
4. Global *X*-coordinate of point *b* on the symmetry axis (the local *z*-axis).
5. Global *Y*-coordinate of point *b* on the symmetry axis (the local *z*-axis).
6. Global *Z*-coordinate of point *b* on the symmetry axis (the local *z*-axis).

Second line:

1. The "word" START.
2. Local *r*-coordinate of the starting point of the line segments.
3. Local *z*-coordinate of the starting point of the line segments.

Third and subsequent data lines define the various line, circular, and parabolic segments (see below for their format) that form the profile of the analytical surface.

Data lines that define the line segments that form the analytical surface for TYPE=SEGMENTS, TYPE=CYLINDER, and TYPE=REVOLUTION:

Data line to define a straight line segment:

1. The "word" LINE.

2. x-coordinate of the end point of the line.
3. y-coordinate of the end point of the line.

Data line to define a circular arc segment (the arc must be less than 179.74°):

1. The "word" CIRCL.
2. x-coordinate of the end point of the circular arc.
3. y-coordinate of the end point of the circular arc.
4. x-coordinate of the center (origin) of the circular arc.
5. y-coordinate of the center (origin) of the circular arc.

Data line to define a parabolic arc segment:

1. The "word" PARAB.
2. Local x-coordinate of the middle point along the parabolic arc.
3. Local y-coordinate of the middle point along the parabolic arc.
4. Local x-coordinate of the end point of the parabolic arc.
5. Local y-coordinate of the end point of the parabolic arc.

For surfaces created with TYPE=SEGMENTS, the x- and y-coordinates are the global X- and Y-coordinates or r- and z-coordinates. For surfaces created with TYPE=CYLINDER, the x- and y-coordinates are the local x- and y-coordinates. For surfaces created with TYPE=REVOLUTION, the x- and y-coordinates are the local r- and z-coordinates.

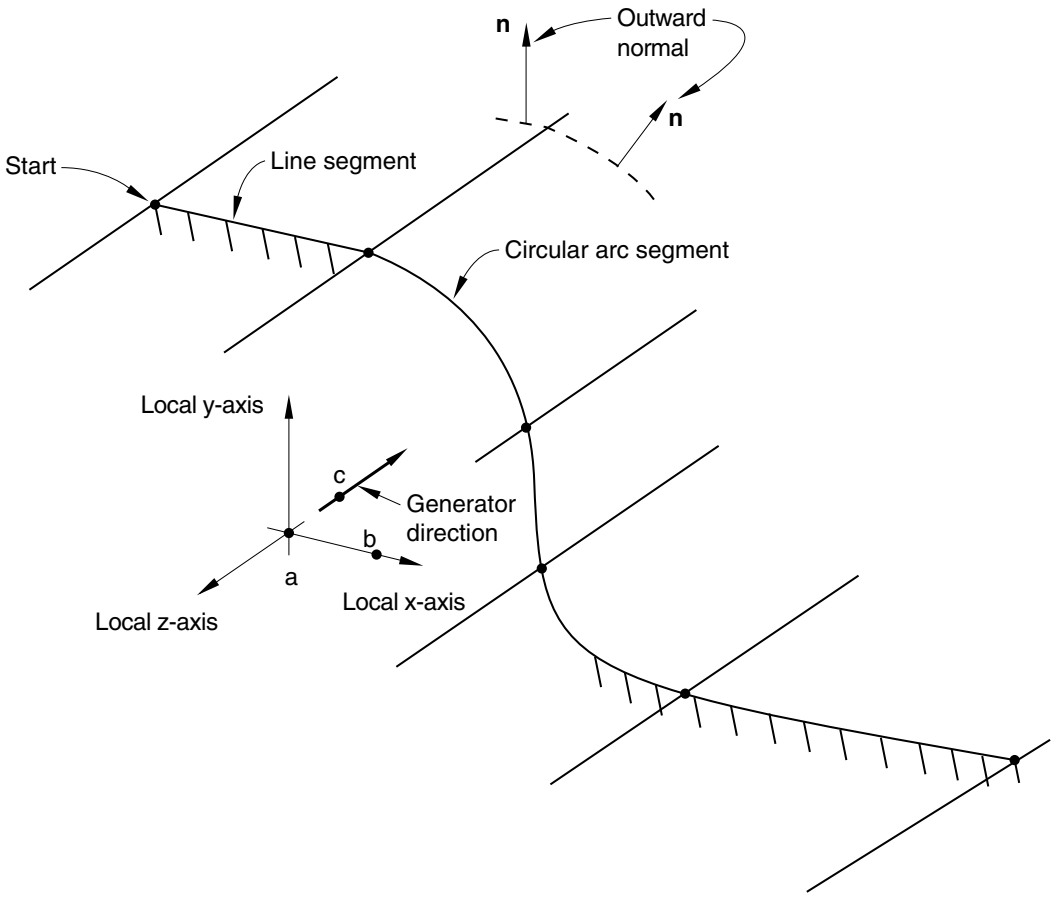


Figure 18.47–1 *SURFACE, TYPE=CYLINDER.

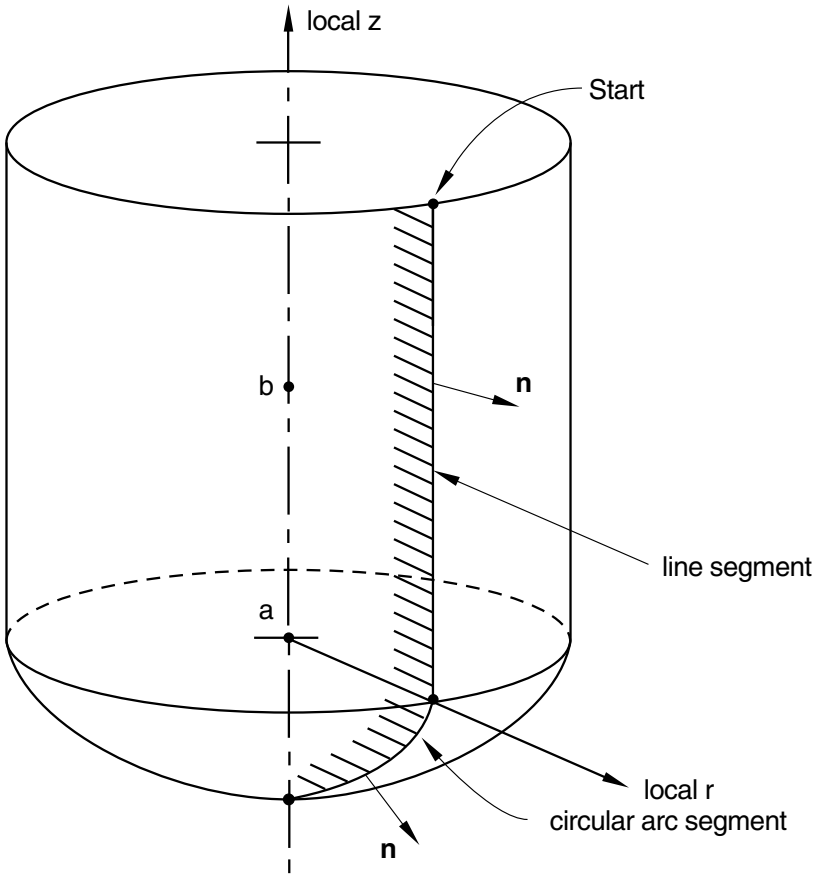


Figure 18.47–2 *SURFACE, TYPE=REVOLUTION.

18.48 *SURFACE BEHAVIOR: Define alternative pressure-overclosure relationships for contact.

This option is used to modify the default hard contact pressure-overclosure relationship in a mechanical contact analysis. Mechanical interactions normal to the surfaces are influenced by this option. It must be used in conjunction with the *SURFACE INTERACTION option or in an Abaqus/Standard analysis with the *GAP option or the *INTERFACE option. By default, Abaqus/Standard will determine whether the contact constraint will be enforced with or without a Lagrange multiplier. You can use the *CONTACT CONTROLS option to override the default.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Part, Part instance, Assembly, Model in Abaqus/Standard; Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- "Mechanical contact properties: overview," Section 33.1.1 of the Abaqus Analysis User's Manual
- "Contact pressure-overclosure relationships," Section 33.1.2 of the Abaqus Analysis User's Manual
- *CONTACT CONTROLS
- *GAP
- *INTERFACE
- *SURFACE INTERACTION

Optional, mutually exclusive parameters:

AUGMENTED LAGRANGE

This parameter applies only to Abaqus/Standard analyses with a default "hard" pressure-overclosure relationship.

Include this parameter to choose the augmented Lagrange method for enforcement of the contact constraint. See "Contact constraint enforcement methods in Abaqus/Standard," Section 34.1.2 of the Abaqus Analysis User's Manual, for a discussion of the default penalty stiffness and penetration tolerance associated with this method (the default penalty stiffness used by this method is often stiffer than that with the straight penalty method). You can specify or modify the penalty stiffness on the data line.

DIRECT

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to choose direct enforcement of contact constraints without approximation or use of augmentation iterations.

PENALTY

This parameter applies only to Abaqus/Standard analyses with a default "hard" pressure-overclosure relationship.

Set PENALTY=LINEAR (default) to choose the linear penalty method for enforcement of the contact constraint. See "Contact constraint enforcement methods in Abaqus/Standard," Section 34.1.2 of the Abaqus Analysis User's Manual, for a discussion of the default linear penalty stiffness. You can specify or modify the penalty stiffness on the data line.

Set PENALTY=NONLINEAR to choose the nonlinear penalty method for enforcement of the contact constraint. See "Contact constraint enforcement methods in Abaqus/Standard," Section 34.1.2 of the Abaqus Analysis User's Manual, for a discussion of the default nonlinear penalty stiffness. You can specify or modify the final nonlinear penalty stiffness and other nonlinear penalty control parameters on the data line.

Optional parameters:

NO SEPARATION

Include this parameter to prevent any separation of the two surfaces once contact has been established.

PRESSURE-OVERCLOSURE

Use this parameter to choose a contact pressure-overclosure relationship other than the default hard contact.

Set PRESSURE-OVERCLOSURE=HARD (default) to choose a pressure-overclosure relationship without physical softening. Note that some numerical softening will occur if a penalty or augmented Lagrange constraint enforcement method is used.

Set PRESSURE-OVERCLOSURE=EXPONENTIAL to define an exponential pressure-overclosure relationship.

Set PRESSURE-OVERCLOSURE=LINEAR to define a linear pressure-overclosure relationship.

Set PRESSURE-OVERCLOSURE=SCALE FACTOR to define a piecewise linear pressure-overclosure relationship based on scaling the default contact stiffness. This option is available only for the general contact algorithm in Abaqus/Explicit.

Set PRESSURE-OVERCLOSURE=TABULAR to define a piecewise linear pressure-overclosure relationship in tabular form.

If a contact area is not defined, such as may occur for node-based surfaces or for GAP- or ITT-type contact elements, "pressure" should be interpreted as force. For contact with three-dimensional beams or trusses, "pressure" should be interpreted as force per unit length.

When used to modify the default surface behavior, the PRESSURE-OVERCLOSURE parameter cannot be used with the NO SEPARATION parameter in an Abaqus/Standard analysis.

Optional data line for AUGMENTED LAGRANGE and PENALTY=LINEAR:

First (and only) line:

1. User-defined linear penalty stiffness. If this field is left blank or is zero, the linear penalty stiffness has the default value or is multiplied by the scale factor given in the third field of the data line.
2. Clearance at which the contact pressure is zero, c_0. The default is zero.
3. Scale factor for the default linear penalty stiffness or for the linear penalty stiffness if specified in the first field of the data line. The default is one.

Optional data line for PENALTY=NONLINEAR:

First (and only) line:

1. User-defined final nonlinear penalty stiffness. If this field is left blank or is zero, the final nonlinear penalty stiffness has the default value or is multiplied by the scale factor given in the third field of the data line.
2. Clearance at which the contact pressure is zero, c_0 (see Figure 18.48–1). The default is zero.
3. Scale factor for the default final nonlinear penalty stiffness or for the final nonlinear penalty stiffness if specified in the first field of the data line. The default is one.
4. Scale factor for the upper quadratic limit d (see Figure 18.48–1), which is equal to the scale factor times the characteristic contact facet length. The default is 0.03.
5. The ratio of initial penalty stiffness over the final penalty stiffness. The default is 0.01.
6. The ratio $(e-c_0)/(d-c_0)$ that defines the lower quadratic limit e (see Figure 18.48–1). The default is 1/3.

Data line for PRESSURE-OVERCLOSURE=EXPONENTIAL:

First (and only) line:

1. Clearance at which the contact pressure is zero, c_0 (see Figure 18.48–2).
2. Pressure at zero clearance, p_0.

The following data item is available only in Abaqus/Explicit analyses:

3. Value of the maximum stiffness, k_{max}. When using penalty contact, large stiffness values obtained from the exponential law may significantly lower the stable time increment size. Choose a suitable maximum stiffness to mitigate this effect (see Figure 18.48–2). By default, k_{max} is set equal to infinity for kinematic contact and the default penalty stiffness for penalty contact.

Data line for PRESSURE-OVERCLOSURE=LINEAR:

First (and only) line:

1. Slope of the pressure-overclosure curve. This value must be positive.

Data line for PRESSURE-OVERCLOSURE=SCALE FACTOR:

First (and only) line:

1. Overclosure factor, r (see Figure 18.48–3), which defines the overclosure measure (used to delineate the segments of the pressure-overclosure curve) as a percentage of the minimum element size in the contact region. This value must be greater than zero. Leave blank if an overclosure measure (second data item) is specified.
2. Overclosure measure, d (see Figure 18.48–3), which defines the overclosure measure (used to delineate the segments of the pressure-overclosure curve) directly. This value must be greater than zero. Leave blank if an overclosure factor (first data item) is specified.
3. Stiffness scale factor, s (see Figure 18.48–3), which defines the geometric scaling of the "base" stiffness and must be greater than one.
4. Initial stiffness scale factor, s_0 (see Figure 18.48–3), which defines an additional scale factor for the "base" default contact stiffness and must be greater than zero. The default value is one.

Data lines for PRESSURE-OVERCLOSURE=TABULAR:

First line:

1. Pressure.
2. Overclosure.

Repeat this data line in ascending order of overclosure value as often as necessary to define the overclosure as a function of pressure. A minimum of two data lines are required. The pressure-overclosure relationship is extrapolated beyond the last overclosure point by continuing the same slope (see Figure 18.48–4).

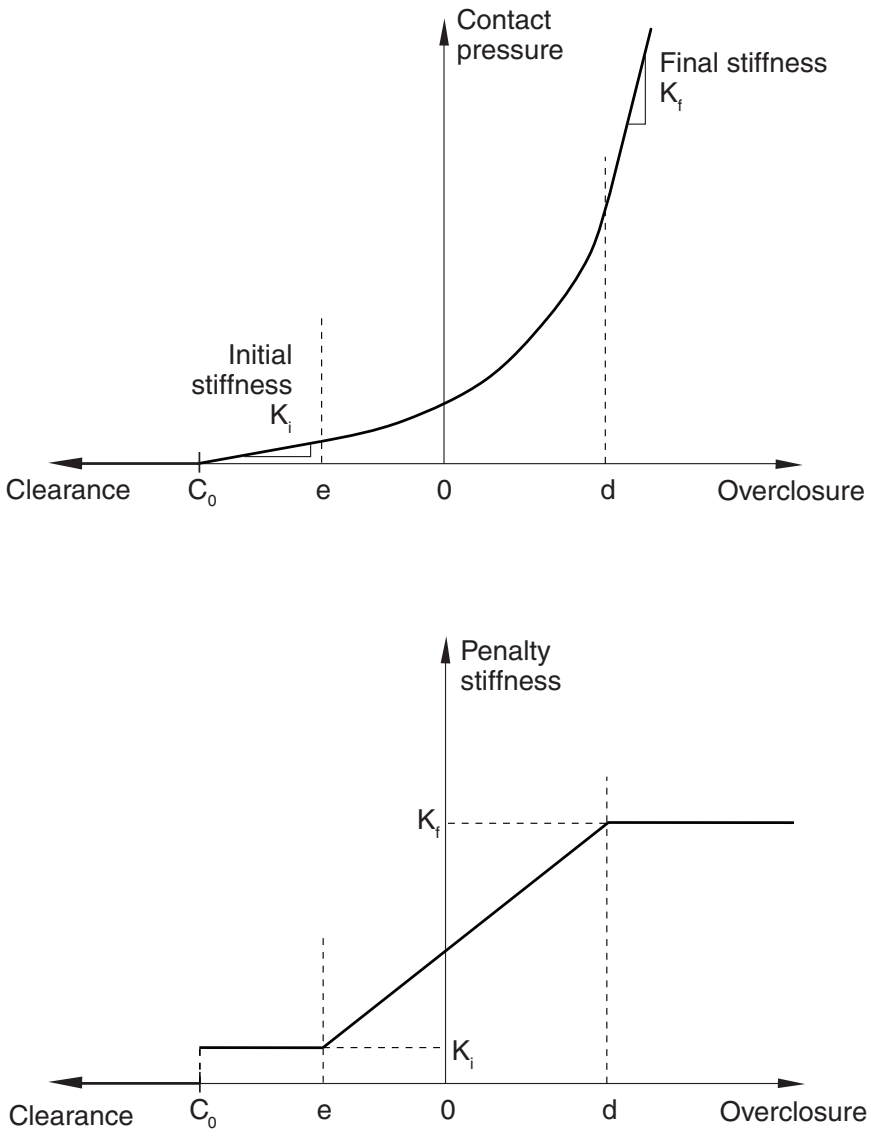


Figure 18.48–1 Nonlinear penalty pressure-overclosure relationship.

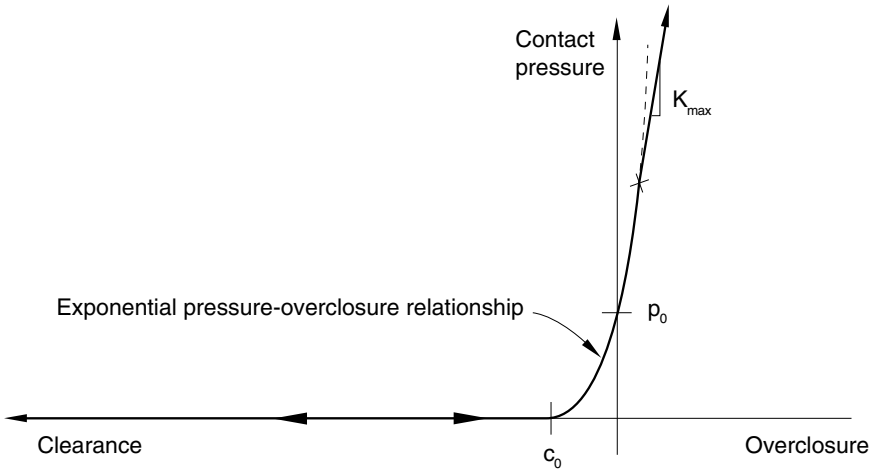


Figure 18.48–2 Exponential pressure-overclosure relationship.

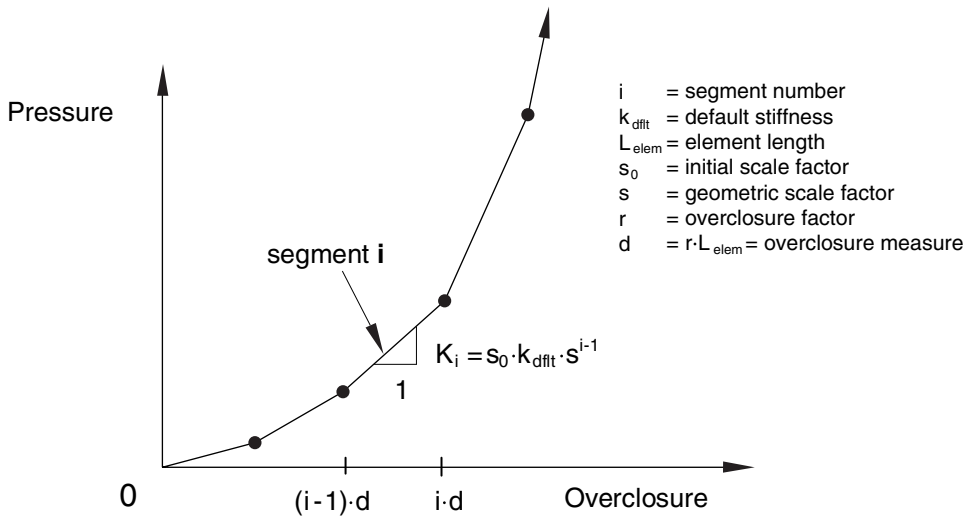


Figure 18.48–3 Scale factor pressure-overclosure relationship.

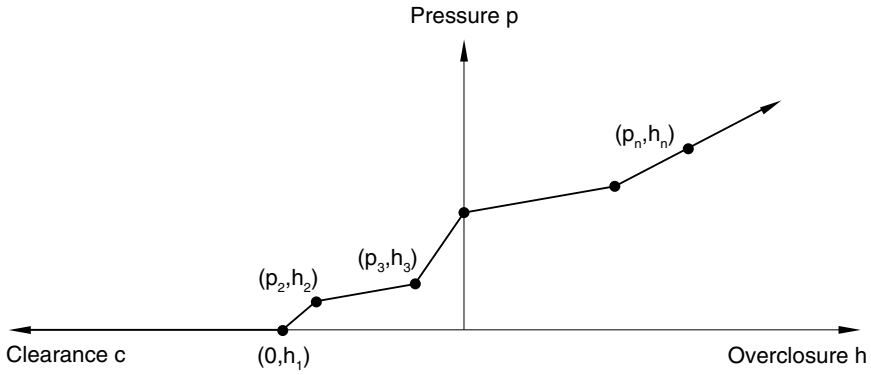


Figure 18.48–4 Pressure-overclosure relationship defined in tabular form.

18.49 *SURFACE FLAW: Define geometry of surface flaws.

This option is used with line spring elements to define the geometry of the part-through crack of the shell.

Product: Abaqus/Standard

Type: Model data

Level: Part, Part instance

Reference:

- "Line spring elements for modeling part-through cracks in shells," Section 29.10.1 of the Abaqus Analysis User's Manual

Required parameter:

SIDE

Set SIDE=POSITIVE or SIDE=NEGATIVE to indicate which surface is cracked.

Optional parameter:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines to define the flaw:

First line:

1. Node number or node set label. This node or node set can be at either side of the crack for element LS6.
2. Crack depth at this location.

Repeat this data line as often as necessary to define the crack depth at all nodal locations along the crack.

18.50 *SURFACE INTERACTION: Define surface interaction properties.

This option is used to create a surface interaction property definition. The surface interaction properties will govern any contact interactions that reference this surface interaction.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Model or Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- "Assigning surface properties for contact pairs in Abaqus/Explicit," Section 32.5.2 of the Abaqus Analysis User's Manual
- "Mechanical contact properties: overview," Section 33.1.1 of the Abaqus Analysis User's Manual
- "Contact pressure-overclosure relationships," Section 33.1.2 of the Abaqus Analysis User's Manual
- "Contact damping," Section 33.1.3 of the Abaqus Analysis User's Manual
- "Frictional behavior," Section 33.1.5 of the Abaqus Analysis User's Manual
- "User-defined interfacial constitutive behavior," Section 33.1.6 of the Abaqus Analysis User's Manual
- "Breakable bonds," Section 33.1.9 of the Abaqus Analysis User's Manual
- "Thermal contact properties," Section 33.2.1 of the Abaqus Analysis User's Manual
- "Electrical contact properties," Section 33.3.1 of the Abaqus Analysis User's Manual
- "Pore fluid contact properties," Section 33.4.1 of the Abaqus Analysis User's Manual
- "UINTER," Section 1.1.34 of the Abaqus User Subroutines Reference Manual
- "VUINTER," Section 1.2.15 of the Abaqus User Subroutines Reference Manual
- "VUINTERACTION," Section 1.2.16 of the Abaqus User Subroutines Reference Manual

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to this surface interaction property.

Use this label on the INTERACTION parameter of the *CONTACT PAIR option (for an Abaqus/Standard contact analysis or an Abaqus/Explicit analysis using the contact pair algorithm) or on the data line of the *CONTACT PROPERTY ASSIGNMENT option (for an Abaqus/Explicit analysis using the general contact algorithm) to assign this surface interaction property to a contact interaction.

Optional parameters:

DEPVAR

This parameter is relevant only when the USER parameter is included.

Set DEPVAR equal to the number of state-dependent variables required for user subroutine **UINTER** in an Abaqus/Standard analysis or for user subroutine **VUINTER** or **VUINTERACTION** in an Abaqus/Explicit analysis. The default is DEPVAR=0.

PAD THICKNESS

This parameter applies only to Abaqus/Explicit analyses using the contact pair algorithm.

Set this parameter equal to the thickness of an interfacial layer between the contacting surfaces. The value can be positive or negative.

PROPERTIES

This parameter is relevant only when the USER parameter is included.

Set this parameter equal to the number of property values needed as data to define the surface interaction model in user subroutine **UINTER** in an Abaqus/Standard analysis or in user subroutine **VUINTER** or **VUINTERACTION** in an Abaqus/Explicit analysis. The default is PROPERTIES=0.

This parameter is ignored when the option is used in association with a connector element.

TRACKING THICKNESS

Set this parameter equal to the thickness that determines the contacting surfaces to be tracked.

In Abaqus/Standard the tracking range will be at least as large as the setting of this parameter; an internally computed default will remain in effect if this parameter setting is less than the default. In Abaqus/Standard this parameter extends the range over which contact opening distance (COPEN) output is provided for separated surfaces.

In Abaqus/Explicit this parameter applies only if user subroutine **VUINTERACTION** is also in effect. Only contacting surfaces whose proximity is within this thickness are available for user-defined interactions. This parameter affects only node-to-surface contact, and the input value for this parameter cannot be negative. An internal default value is used if a zero value is input or if the parameter is omitted.

UNSYMM

This parameter applies only to Abaqus/Standard analyses in which the USER parameter is included.

Include this parameter when the interface stiffness matrix is not symmetric. This parameter causes Abaqus/Standard to use its unsymmetric equation solution procedures.

USER

In an Abaqus/Standard analysis, include this parameter if the surface interaction model is to be defined in user subroutine **UINTER**

In an Abaqus/Explicit analysis, include this parameter without any assignment if the surface interaction model is to be defined in user subroutine **VUINTER**. Set USER=INTERACTION if the surface interaction model is to be defined in user subroutine **VUINTERACTION**. **VUINTER** is applicable to contact pairs, whereas **VUINTERACTION** is applicable to general contact.

When this parameter is included, the *SURFACE BEHAVIOR option and its various suboptions cannot be used under the same interaction definition.

Optional data line for two-dimensional models in Abaqus/Standard or for contact pairs involving node-based surfaces in Abaqus/Standard, if the USER parameter is omitted:

First (and only) line:

1. Out-of-plane thickness of the surface or cross-sectional area for every node in the node-based surface. The default value is 1.0.

Data lines to define the surface interaction in an Abaqus/Standard analysis if the USER parameter is used:

First line:

1. Out-of-plane thickness of the surface for two-dimensional models or cross-sectional area for every node in the node-based surface. The default value for both is 1.0. In other situations, enter a blank line.

Second line (needed only if the PROPERTIES parameter is used):

1. Enter the values of the surface interaction properties, eight per line.

Repeat this data line as often as necessary to define all material constants.

Data lines to define the surface interaction in an Abaqus/Explicit analysis if the PROPERTIES parameter is used:

First line:

1. Enter a blank line.

Second line:

1. Enter the values of the surface interaction properties, eight per line.

Repeat this data line as often as necessary to define all material constants.

18.51 *SURFACE PROPERTY: Define surface properties for cavity radiation.

This option is used to define surface properties for cavity radiation analysis. It must immediately precede the *EMISSIVITY option.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

References:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual
- *EMISSIVITY

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the property name for cross-referencing to the surface definitions for which properties are being defined. Only element-based surfaces can be used.

There are no data lines associated with this option.

18.52 *SURFACE PROPERTY ASSIGNMENT: Assign surface properties to a surface for the general contact algorithm.

This option is used to modify surface properties for surfaces that are involved in general contact interactions. It must be used in conjunction with the *CONTACT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data in Abaqus/Standard; Model or history data in Abaqus/Explicit

Level: Model in Abaqus/Standard; Model or Step in Abaqus/Explicit

Abaqus/CAE: Interaction module

References:

- "Surface properties for general contact in Abaqus/Standard," Section 32.2.2 of the Abaqus Analysis User's Manual
- "Assigning surface properties for general contact in Abaqus/Explicit," Section 32.4.2 of the Abaqus Analysis User's Manual
- *CONTACT

Required parameter:

PROPERTY

Use this parameter to specify the property type being assigned. To modify more than one type of surface property, use this option more than once with different values for the PROPERTY parameter.

Set PROPERTY=FEATURE EDGE CRITERIA to control which primary feature edges and secondary feature edges should be activated in the general contact domain. This parameter setting is available only in Abaqus/Explicit.

Set PROPERTY=GEOMETRIC CORRECTION to assign geometric corrections. This parameter setting is available only in Abaqus/Standard.

Set PROPERTY=OFFSET FRACTION to assign the surface offset as a fraction of the surface thickness.

Set PROPERTY=THICKNESS to assign the surface thickness.

Data lines for PROPERTY=FEATURE EDGE CRITERIA:

First line:

1. Surface name. A non-blank surface name is required if the ALL EDGES or PICKED EDGES options are specified. If the surface name is omitted when using the PERIMETER EDGES, NO FEATURE EDGES, or cutoff feature angle options, a default surface that encompasses the entire general contact domain is assumed.

2. The criterion for primary feature edges. The "words" PERIMETER EDGES (default), the "words" ALL EDGES, the "words" PICKED EDGES, the "words" NO FEATURE EDGES, or a scalar value representing the cutoff feature angle (in degrees).
3. The criterion for secondary feature edges. The "words" ALL REMAINING EDGES, the "words" PERIMETER EDGES, the "words" PICKED EDGES, or a scalar value representing the cutoff feature angle (in degrees).

Repeat this data line as often as necessary. If the feature edge criteria assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=GEOMETRIC CORRECTION to define smoothing on regions of two-dimensional surfaces that correspond (or nearly correspond) to a circular arc:

First line:

1. Surface name.
2. The "word" CIRCUMFERENTIAL.
3. Global *X*-coordinate of point *a*, the approximate center (origin) of the circular arc (see Figure 18.54–1).
4. Global *Y*-coordinate of point *a*, the approximate center (origin) of the circular arc.

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region.

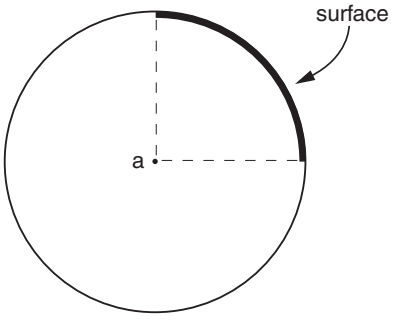


Figure 18.52–1 Two-dimensional circumferential smoothing.

Data lines for PROPERTY=GEOMETRIC CORRECTION to define smoothing on regions of surfaces that correspond (or nearly correspond) to a surface of revolution:

First line:

1. Surface name.
2. The "word" CIRCUMFERENTIAL.

3. Global *X*-coordinate of point *a* on the approximate axis of revolution for the surface (see Figure 18.54–2).
4. Global *Y*-coordinate of point *a* on the approximate axis of revolution for the surface.
5. Global *Z*-coordinate of point *a* on the approximate axis of revolution for the surface.
6. Global *X*-coordinate of point *b* on the approximate axis of revolution for the surface (see Figure 18.54–2).
7. Global *Y*-coordinate of point *b* on the approximate axis of revolution for the surface.
8. Global *Z*-coordinate of point *b* on the approximate axis of revolution for the surface.

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region.

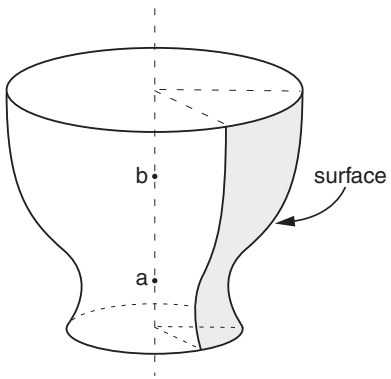


Figure 18.52–2 Three-dimensional circumferential smoothing.

Data lines for PROPERTY=GEOMETRIC CORRECTION to define smoothing on regions of surfaces that correspond (or nearly correspond) to a spherical section:

First line:

1. Surface name.
2. The "word" SPHERICAL.
3. Global *X*-coordinate of point *a*, the approximate center (origin) of the sphere (see Figure 18.54–3).
4. Global *Y*-coordinate of point *a*, the approximate center (origin) of the sphere.
5. Global *Z*-coordinate of point *a*, the approximate center (origin) of the sphere.

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region.

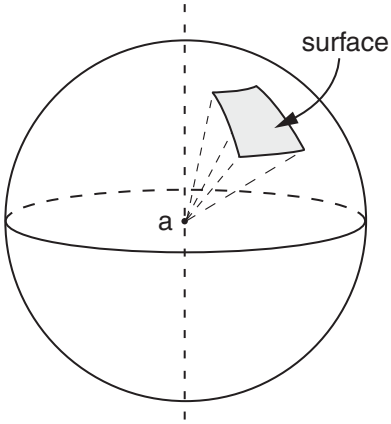


Figure 18.52–3 Spherical smoothing.

Data lines for PROPERTY=GEOMETRIC CORRECTION to define surface regions that should not be smoothed (corresponds to default):

First line:

1. Surface name.
2. Leave blank or specify the "word" NONE.

Repeat this data line as often as necessary. If the geometry correction assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=OFFSET FRACTION:

First line:

1. Surface name. If the surface name is omitted, a default surface that encompasses the entire general contact domain is assumed. Faces specified on elements other than shell elements, membrane elements, rigid elements, and surface elements will be ignored.
2. The "word" ORIGINAL (default), the "word" SPOS, the "word" SNEG, or a value between –0.5 and 0.5. The offset defines the distance (as a fraction of the thickness) from the midsurface to the reference surface (containing the nodes of the element). Positive values of the offset are in the positive element normal direction. The default is ORIGINAL, which indicates that the offset specified in element section definitions (via the OFFSET parameter on *SHELL SECTION or *RIGID BODY) will be used.

Repeat this data line as often as necessary. If the offset fraction assignments overlap, the last assignment applies in the overlap region.

Data lines for PROPERTY=THICKNESS:

First line:

1. Surface name. If the surface name is omitted, a default surface that encompasses the entire general contact domain is assumed.
2. The "word" ORIGINAL (default), the "word" THINNING (available only in Abaqus/Explicit), or a scalar value representing a nominal thickness.
3. A constant scaling factor (default is 1.0).

Repeat this data line as often as necessary. If the thickness assignments overlap, the last assignment applies in the overlap region.

18.53 *SURFACE SECTION: Specify section properties for surface elements.

This option is used to specify a surface element cross-section. It must be used in conjunction with the *REBAR LAYER option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- "Surface elements," Section 29.7.1 of the Abaqus Analysis User's Manual
- "Defining reinforcement," Section 2.2.3 of the Abaqus Analysis User's Manual
- *REBAR LAYER

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the surface elements for which the section properties are being defined.

Optional parameter:

DENSITY

Set this parameter equal to the mass density per unit area of the surface element section.

There are no data lines associated with this option.

18.54 *SURFACE SMOOTHING: Define surface smoothing methods.

This option is used to create a surface smoothing definition for contact interactions. It must be used in conjunction with the *CONTACT PAIR option. The defined smoothing methods apply to the specified regions of the surfaces in the referenced contact pair.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Interaction module

Reference:

- "Smoothing contact surfaces in Abaqus/Standard," Section 34.1.3 of the Abaqus Analysis User's Manual

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to this surface smoothing definition.

This label is referred to by the GEOMETRIC CORRECTION parameter on the *CONTACT PAIR option.

Data lines to define the surface regions on which smoothing is applied:

Data line to define smoothing on regions of two-dimensional surfaces that correspond (or nearly correspond) to a circular arc (see Figure 18.54–1):

1. Name of slave surface or surface corresponding to a subregion of the slave surface.
2. Name of master surface or surface corresponding to a subregion of the master surface.
3. The "word" CIRCUMFERENTIAL.
4. Global X-coordinate of point a, the approximate center (origin) of the circular arc.
5. Global Y-coordinate of point a, the approximate center (origin) of the circular arc.

Data line to define smoothing on regions of surfaces that correspond (or nearly correspond) to a surface of revolution (see Figure 18.54–2):

1. Name of slave surface or surface corresponding to a subregion of the slave surface.
2. Name of master surface or surface corresponding to a subregion of the master surface.
3. The "word" CIRCUMFERENTIAL.
4. Global X-coordinate of point a on the approximate axis of revolution for the surface.

5. Global *Y*-coordinate of point *a* on the approximate axis of revolution for the surface.
6. Global *Z*-coordinate of point *a* on the approximate axis of revolution for the surface.
7. Global *X*-coordinate of point *b* on the approximate axis of revolution for the surface (see Figure 18.54–2).
8. Global *Y*-coordinate of point *b* on the approximate axis of revolution for the surface.
9. Global *Z*-coordinate of point *b* on the approximate axis of revolution for the surface.

Data line to define smoothing on regions of surfaces that correspond (or nearly correspond) to a sphere section (see Figure 18.54–3):

1. Name of slave surface or surface corresponding to a subregion of the slave surface.
2. Name of master surface or surface corresponding to a subregion of the master surface.
3. The "word" SPHERICAL.
4. Global *X*-coordinate of point *a*, the approximate center (origin) of the sphere.
5. Global *Y*-coordinate of point *a*, the approximate center (origin) of the sphere.
6. Global *Z*-coordinate of point *a*, the approximate center (origin) of the sphere.

Repeat the above data lines as often as necessary to define all surface regions that require smoothing.

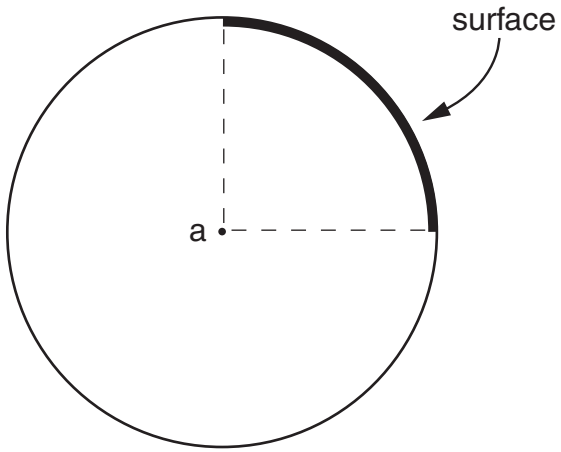


Figure 18.54–1 Two-dimensional circumferential smoothing.

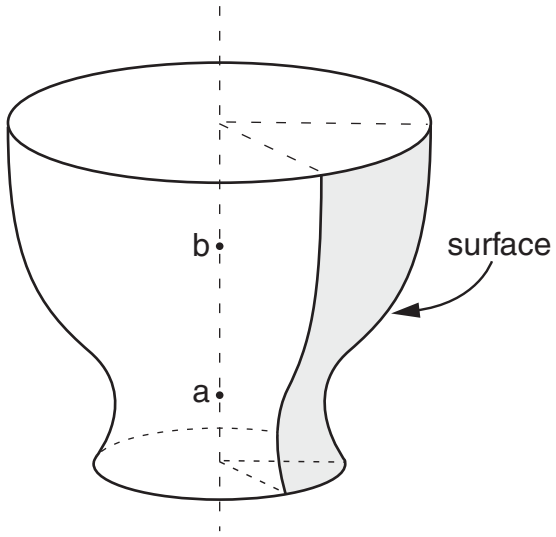


Figure 18.54–2 Three-dimensional circumferential smoothing.

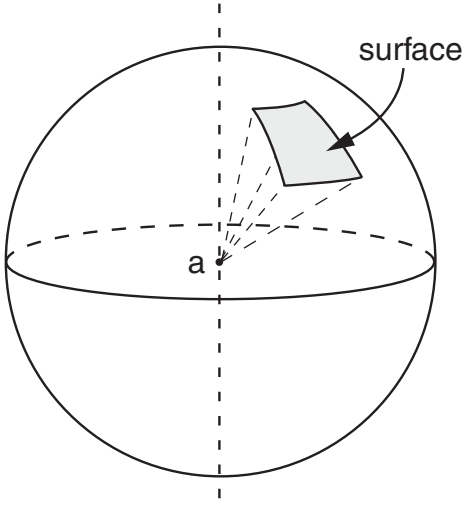


Figure 18.54–3 Spherical smoothing.

18.55 *SWELLING: Specify time-dependent volumetric swelling.

This option is used to specify time-dependent metal swelling for a material. Swelling behavior defined by this option is active only during *SOILS, CONSOLIDATION; *COUPLED TEMPERATURE-DISPLACEMENT; and *VISCO procedures.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual
- "CREEP," Section 1.1.1 of the Abaqus User Subroutines Reference Manual

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the volumetric swelling strain rate in addition to temperature. If this parameter is omitted, it is assumed that the volumetric swelling strain rate is constant or depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

LAW

Set LAW=INPUT (default) to define the swelling behavior on the data lines.
Set LAW=USER to define the swelling behavior in user subroutine **CREEP**.

Data lines for LAW=INPUT:

First line:

1. Volumetric swelling strain rate.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of volumetric swelling strain rate on temperature and other predefined field variables.

18.56 *SYMMETRIC MODEL GENERATION: Create a three-dimensional model from an axisymmetric or partial three-dimensional model.

This option is used to create a three-dimensional model by revolving the cross-section of an axisymmetric model about a symmetry axis, by revolving a single three-dimensional sector about the symmetry axis, or by combining two parts of a symmetric three-dimensional model where one part is the original model and the other part is obtained by reflecting the original part through a line or a plane.

Product: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

Reference:

- "Symmetric model generation," Section 10.4.1 of the Abaqus Analysis User's Manual

Required, mutually exclusive parameters:

PERIODIC

Use this parameter to indicate that a periodic three-dimensional model must be generated by revolving a single three-dimensional sector of the model about the symmetry axis (see Figure 18.56–1).

Set PERIODIC=CONSTANT (default) to indicate that each generated sector in the periodic model has a constant angle.

Set PERIODIC=VARIABLE to indicate that each generated sector in the periodic model can have a variable angle in the circumferential direction.

In both cases each sector always has the same geometry and mesh. The surfaces on both sides of the original sector must be completely planar when PERIODIC=VARIABLE. If the surface meshes on either side of the original sector are not matched completely, constraints between the neighboring pairs of corresponding surfaces specified on the data lines will be applied with an automatically generated *TIE option when the periodic three-dimensional model is generated.

REFLECT

Set REFLECT=LINE to indicate that a three-dimensional model must be generated by reflecting a partial three-dimensional model through a symmetry line (Figure 18.56–3).

Set REFLECT=PLANE to indicate that a three-dimensional model must be generated by reflecting a partial three-dimensional model through a symmetry plane (Figure 18.56–4).

REVOLVE

Include this parameter to indicate that a three-dimensional model must be generated by revolving the cross-section of an axisymmetric mesh about the symmetry axis. See Figure 18.56–2.

Optional parameters:

ELEMENT OFFSET

Set this parameter equal to an integer to define the offset for element numbering. When the REVOLVE parameter is used, the offset is added to each element number on the previous cross-section to obtain the numbering of the elements on the next cross-section, starting at the reference cross-section, $\theta = 0.0$. The reference cross-section uses the same numbering as the original axisymmetric model. When the REFLECT parameter is used, the offset is added to the original element numbers to define the numbering on the reflected part. The default and minimum value is the largest element number used in the original model.

FILE NAME

Set this parameter equal to the name of an external file (without an extension) to which keyword and data lines for the model definition will be written. The extension **`.axi`** will be added to the file name provided by the user. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names.

NODE OFFSET

Set this parameter equal to an integer to define the offset used for node numbering. When the REVOLVE parameter is used, the offset is added to each node number on the previous cross-section to obtain the numbering of the nodes on the next cross-section, starting at the reference cross-section, $\theta = 0.0$. The reference cross-section uses the same numbering as the original axisymmetric model. When the REFLECT parameter is used, the offset is added to the original node numbers to define the numbering on the reflected part. The default and minimum value is the largest node number used in the original model.

TOLERANCE

Set this parameter equal to the distance to be used in the search for duplicate nodes. Duplicate nodes on the axis of revolution of a revolved model, on the connection planes between sectors of a periodic model, and on the connection plane between the two parts of a reflected model will be eliminated. The default distance is 1.0% of the average element dimension.

Data lines if each generated sector in the periodic model has a constant angle (PERIODIC=CONSTANT):

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. Segment angle, θ (in degrees), of the original three-dimensional sector. $0.0^\circ < \theta \leq 360.0^\circ$.

2. Number of three-dimensional repetitive sectors, including the original sector in the generated periodic model. The default is 1.

Third line (needed if the surface meshes on either side of the original sector are not matched completely):

1. The surface name on one side of the original sector.
2. The corresponding surface name on the other side of the original sector, measured at a positive angle, θ, around the revolving axis.
3. Tolerance distance within which nodes on one surface of a sector must lie from the corresponding surface of the neighboring sector to be constrained. Nodes on the surface of the sector that are further away from the corresponding surface of the neighboring sector than this distance are not constrained. The default value for this tolerance distance is 5% or 10% of the typical element size in the surfaces of the original sector, depending on whether node-to-surface or surface-to-surface type of constraints are used, respectively.
4. Include the "word" SURFACE (default) to generate a surface-to-surface type of constraint or the "word" NODE to generate a node-to-surface type of constraint.

Repeat the third data line as often as necessary to define pairs of corresponding surfaces on each side of the original repetitive sector. Constraints between the neighboring pairs of corresponding surfaces will be applied with an automatically generated *TIE option when the periodic three-dimensional model is generated.

Data lines if each generated sector in the periodic model has a variable angle (PERIODIC=VARIABLE):

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. Segment angle, θ (in degrees), of the original three-dimensional sector. $0.0^\circ < \theta \leq 360.0^\circ$.
2. Number of three-dimensional sectors that have the same angle as the original sector to be generated, including the original sector. The default is 1.

Third line:

1. Number of additional three-dimensional sectors to be generated.
2. Angular scaling factor in the circumferential direction with respect to the original sector. The default is 1.0.

Repeat the third data line as often as necessary to define all the sectors of the model in the circumferential direction.

Subsequent lines (needed if the surface meshes on either side of the original sector are not matched completely):

1. The surface name on one side of the original sector.
2. The corresponding surface name on the other side of the original sector, measured at a positive angle, θ, around the revolving axis.
3. Tolerance distance within which nodes on one surface of a sector must lie from the corresponding surface of the neighboring sector to be constrained. Nodes on the surface of the sector that are further away from the corresponding surface of the neighboring sector than this distance are not constrained. The default value for this tolerance distance is 5% or 10% of the typical element size in the surfaces of the original sector, depending on whether node-to-surface or surface-to-surface type of constraints are used, respectively.
4. Include the "word" SURFACE (default) to generate a surface-to-surface type of constraint or the "word" NODE to generate a node-to-surface type of constraint.

Repeat the subsequent data line as often as necessary to define more pairs of corresponding surfaces on each side of the original sector. Constraints between the neighboring pairs of corresponding surfaces will be applied with the automatically generated *TIE option when the periodic three-dimensional model is generated.

Data line if REFLECT=LINE:

First (and only) line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Data lines if REFLECT=PLANE:

First line:

1. *X*-coordinate of point *a*.
2. *Y*-coordinate of point *a*.
3. *Z*-coordinate of point *a*.
4. *X*-coordinate of point *b*.
5. *Y*-coordinate of point *b*.
6. *Z*-coordinate of point *b*.

Second line:

1. X-coordinate of point c.
2. Y-coordinate of point c.
3. Z-coordinate of point c.

Data lines if the REVOLVE parameter is included:

First line:

1. X-coordinate of point a.
2. Y-coordinate of point a.
3. Z-coordinate of point a.
4. X-coordinate of point b.
5. Y-coordinate of point b.
6. Z-coordinate of point b.

Second line:

1. X-coordinate of point c.
2. Y-coordinate of point c.
3. Z-coordinate of point c.

Third line:

1. Segment angle, θ, through which the cross-section must be revolved. $0.0° < \theta \leq 360.0°$. The segments are connected, so, except for the first segment, each segment starts at the end point of the segment given on the previous data line.
2. Number of subdivisions or elements to be used in the segment. The default is 1. A single element subdivision must not exceed 45° when general elements are used or 180° when cylindrical elements are used.
3. Bias ratio to be used in the spacing of nodes generated over the segment. The value is set to the ratio of adjacent angles between nodes along each arc of nodes generated. Thus, if the value is less than one, the nodes are concentrated toward the beginning of the segment; and if the value is greater than one, the nodes are concentrated toward the end of the segment. The default is 1.0.
4. Include the "word" GENERAL (default) to generate general three-dimensional elements or the "word" CYLINDRICAL to generate cylindrical elements.

Repeat the third data line as often as necessary to define the discretization of the model in the circumferential direction.

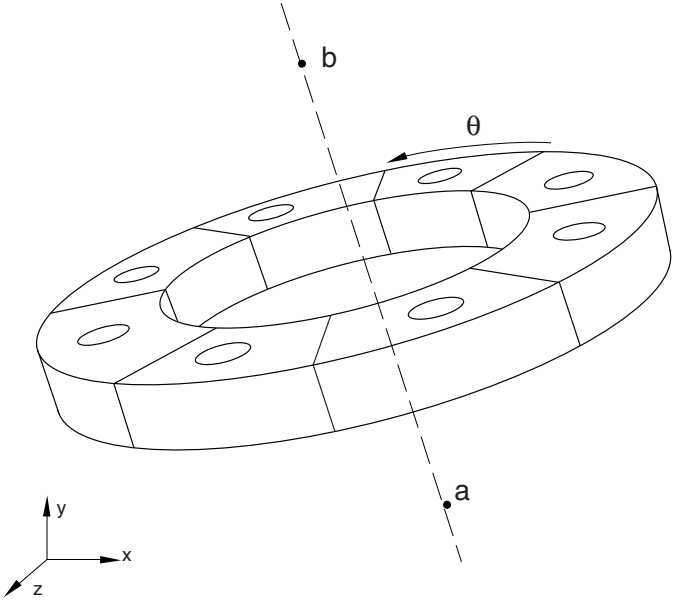


Figure 18.56–1 Revolving a single three-dimensional repetitive sector to create a periodic structure.

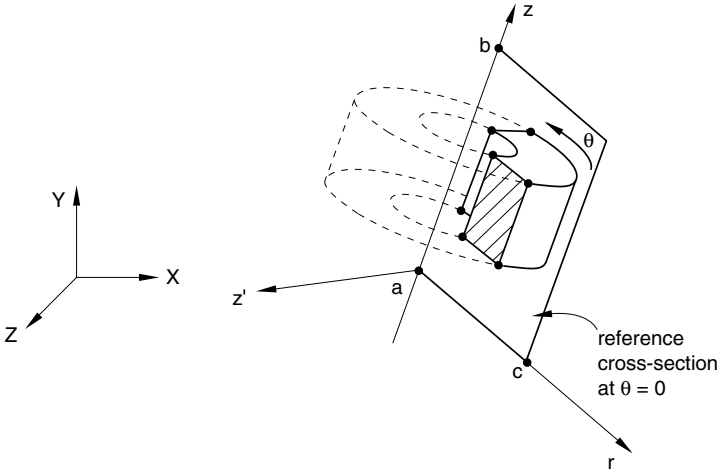


Figure 18.56–2 Revolving an axisymmetric cross-section.

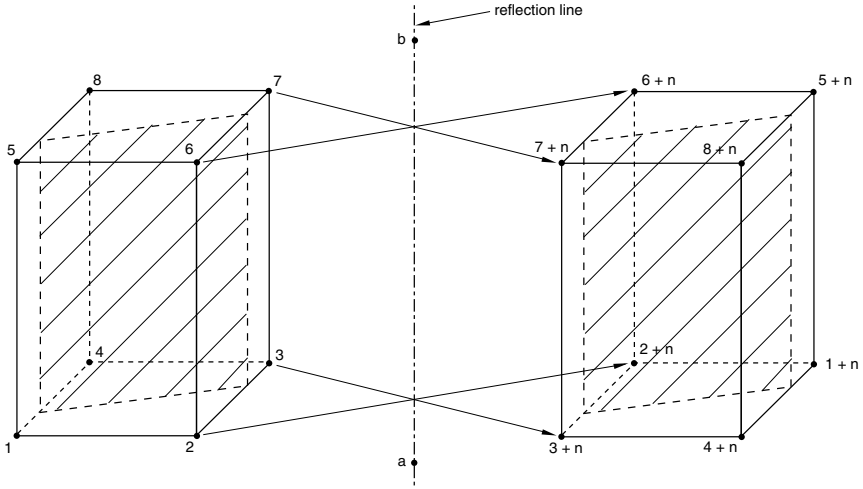


Figure 18.56–3 Reflecting a three-dimensional model through line *ab* with node offset *n*.

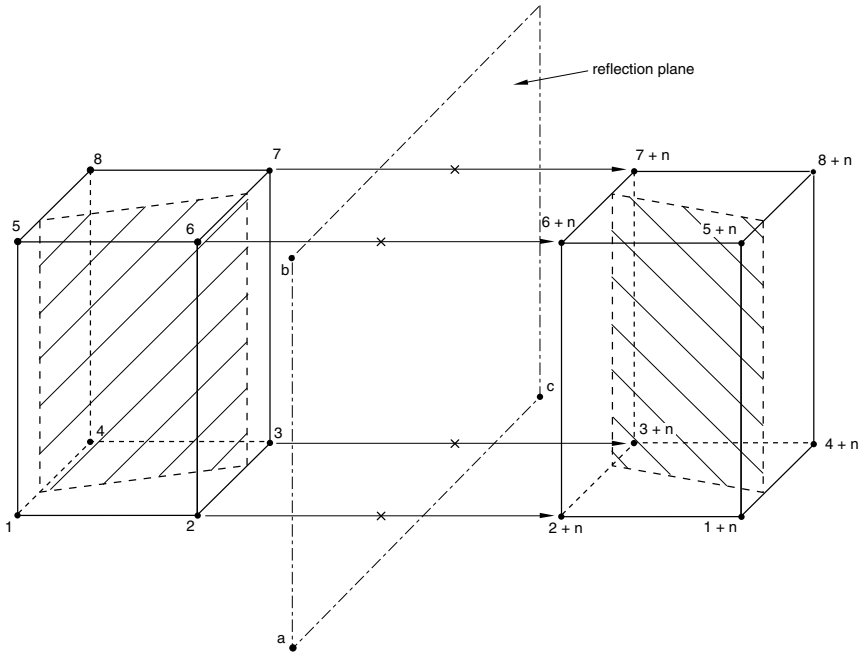


Figure 18.56–4 Reflecting a three-dimensional model through a plane *abc* with node offset *n*.

18.57 *SYMMETRIC RESULTS TRANSFER: Import results from an axisymmetric or partial three-dimensional analysis.

This option is used to transfer a solution from an axisymmetric analysis to a three-dimensional model or to transfer the solution of a partial three-dimensional model to a full three-dimensional model. It can be used only in conjunction with the *SYMMETRIC MODEL GENERATION option.

Product: Abaqus/Standard

Type: Model data

Level: This option is not supported in a model defined in terms of an assembly of part instances.

References:

- "Transferring results from a symmetric mesh or a partial three-dimensional mesh to a full three-dimensional mesh," Section 10.4.2 of the Abaqus Analysis User's Manual
- "Symmetric model generation," Section 10.4.1 of the Abaqus Analysis User's Manual
- *SYMMETRIC MODEL GENERATION

Optional parameters:

STEP

Set this parameter equal to the step number on the restart file at which the results must be obtained. If this parameter is omitted, the results will be obtained at the last available step found on the restart file.

INC

Set this parameter equal to the increment number on the restart file at which the results must be obtained. If this parameter is omitted, the results will be obtained at the end of the step specified by the STEP parameter.

ITERATION

This parameter is relevant only if the solution is transferred from a previous direct cyclic analysis.

Set this parameter equal to the iteration number on the restart file at which the results must be obtained. Since restart information can be written only at the end of an iteration in a direct cyclic analysis, the INC parameter is irrelevant and is ignored if the ITERATION parameter is specified.

If this parameter is omitted, the results will be obtained at the end of the step specified by the STEP parameter.

UNBALANCED STRESS

Set UNBALANCED STRESS=STEP (default) if the stress unbalance is to be resolved in the first increment.

Set UNBALANCED STRESS=RAMP if the stress unbalance is to be resolved linearly over the step.

There are no data lines associated with this option.

18.58 *SYSTEM: Specify a local coordinate system in which to define nodes.

This option is used to define nodes by accepting coordinates relative to a specified local rectangular coordinate system and generating the nodal coordinates in the global coordinate system.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Not applicable; instancing a part in the Assembly module creates a local coordinate system.

Reference:

- "Node definition," Section 2.1.1 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data lines to define a local coordinate system:

First line:

1. X_a, global *X*-coordinate of the origin of the local coordinate system (point *a* in Figure 18.58–1).
2. Y_a, global *Y*-coordinate of the origin of the local coordinate system.
3. Z_a, global *Z*-coordinate of the origin of the local coordinate system.

The following entries are not needed for a pure translation:

4. X_b, global *X*-coordinate of a point on the X^1-axis of the local coordinate system (point *b* in Figure 18.58–1).
5. Y_b, global *Y*-coordinate of a point on the X^1-axis of the local coordinate system.
6. Z_b, global *Z*-coordinate of a point on the X^1-axis of the local coordinate system.

Second line (optional; if not provided, the Z-axis direction remains unchanged, and the X^1-axis is projected onto the (X, Y) plane):

1. X_c, global *X*-coordinate of a point in the (X^1, Y^1) plane of the local coordinate system, on the side of the positive Y^1-axis (for example, point *c* in Figure 18.58–1).
2. Y_c, global *Y*-coordinate of a point in the (X^1, Y^1) plane of the local coordinate system, on the side of the positive Y^1-axis.
3. Z_c, global *Z*-coordinate of a point in the (X^1, Y^1) plane of the local coordinate system, on the side of the positive Y^1-axis.

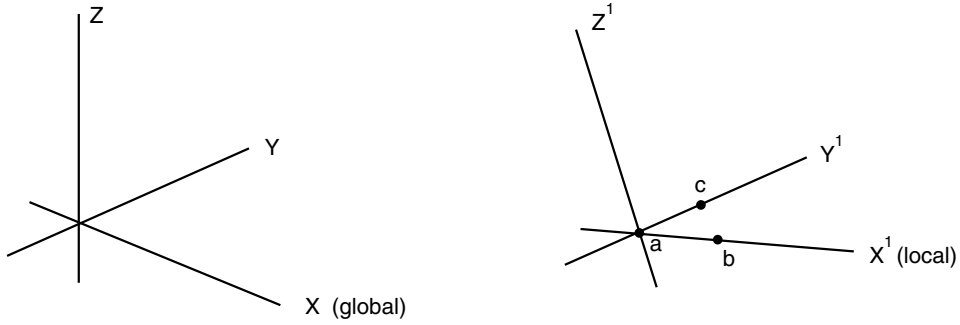


Figure 18.58–1 Local coordinate system.

19. T

19.1 *TEMPERATURE: Specify temperature as a predefined field.

This option is used to specify temperature as a predefined field during an analysis.

To use this option in a restart analysis of Abaqus/Standard, either *TEMPERATURE or *INITIAL CONDITIONS, TYPE=TEMPERATURE must have been specified in the original analysis.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Load module

References:

- "Predefined fields," Section 30.6.1 of the Abaqus Analysis User's Manual
- "UTEMP," Section 1.1.46 of the Abaqus User Subroutines Reference Manual

Optional parameters for using the data line format:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve that gives the time variation of the temperature throughout the step (see "Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual).

If this parameter is omitted in an Abaqus/Standard analysis, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option (see "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted in an Abaqus/Explicit analysis, a linear interpolation is done over the step.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

OP

Set OP=MOD (default) for existing *TEMPERATURE values to remain, with this option modifying existing temperatures or defining additional temperatures.

Set OP=NEW if all existing *TEMPERATURE values should be removed. New temperatures can be defined.

For a general analysis step, a temperature that is removed via OP=NEW is reset to the value given on the *INITIAL CONDITIONS option or to zero if no initial temperature was defined. For a linear perturbation step, a temperature that is removed via OP=NEW is always reset to zero. If temperatures are being returned to their initial condition values, the AMPLITUDE parameter

described above does not apply. Rather, the AMPLITUDE parameter given on the *STEP option governs the behavior in an Abaqus/Standard analysis, and the temperatures are always ramped back to their initial conditions in Abaqus/Explicit analyses. If temperatures are being reset to new values (not to initial conditions) via OP=NEW, the AMPLITUDE parameter described above applies.

Required parameter for reading temperatures from the results or output database file:

FILE

Set this parameter equal to the name of the results or output database file from which the data are read. The file extension is optional; however, if both **`.fil`** and **`.odb`** files exist, the results file will be used if the INTERPOLATE parameter is omitted. If the INTERPOLATE parameter is used, an output database file must exist. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. This parameter cannot be used in a *STATIC, RIKS analysis step.

Optional parameters for reading temperatures from the results or output database file:

ABSOLUTE EXTERIOR TOLERANCE

This parameter is relevant only for use with the INTERPOLATE parameter. Set this parameter equal to the absolute value (given in the units used in the model) by which nodes of the current model may lie outside the region of the model in the output database specified by the FILE parameter. If this parameter is not used or has a value of 0.0, the EXTERIOR TOLERANCE parameter will apply.

EXTERIOR TOLERANCE

This parameter is relevant only for use with the INTERPOLATE parameter. Set this parameter equal to the fraction of the average element size by which nodes of the current model may lie outside the region of the elements of the model in the output database specified by the FILE parameter. The default value is 0.05.

If both tolerance parameters are specified, Abaqus uses the tighter tolerance.

BSTEP

Set this parameter equal to the step number (of the analysis whose results or output database file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus will begin reading temperature data from the first step available on the file read.

BINC

Set this parameter equal to the increment number (of the analysis whose results or output database file is being used as input to this option) that begins the history data to be read. If no value is supplied, Abaqus will begin reading temperature data from the first increment available (excluding any zero increments if the results file was written in Abaqus/Standard using *FILE FORMAT, ZERO INCREMENT) for step BSTEP on the results or output database file.

ESTEP

Set this parameter equal to the step number (of the analysis whose results or output database file is being used as input to this option) that ends the history data to be read. If no value is supplied, ESTEP is taken as equal to BSTEP.

EINC

Set this parameter equal to the increment number (of the analysis whose results or output database file is being used as input to this option) that ends the history data to be read. If no value is supplied, EINC is taken as the last available increment of step ESTEP on the file read.

BTRAMP

Set this parameter equal to the starting time (measured relative to the total step time period) after which the temperatures read from the results file will be ramped to their initial condition values. The default value is 1.0, in which case no temperature ramping takes place. This feature is used to create a cyclic temperature history from a prior heat transfer analysis that is not cyclic.

INTERPOLATE

Include this parameter to indicate that the temperature field needs to be interpolated between dissimilar meshes. This feature is used to read temperatures from an output database file generated during a heat transfer analysis or generated during a global model analysis used with the submodeling capability. This parameter and the MIDSIDE parameter are mutually exclusive. If the heat transfer analysis uses first-order elements and the current mesh is the same but uses second-order elements, use the MIDSIDE parameter instead.

MIDSIDE

Include this parameter to indicate that midside node temperatures in second-order elements are to be interpolated from corner node temperatures. This feature is used to read temperatures from a results or an output database file generated during a heat transfer analysis using first-order elements. This parameter and the INTERPOLATE parameter are mutually exclusive.

Required parameter for defining data in user subroutine `UTEMP`:

USER

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to indicate that user subroutine **`UTEMP`** will be used to define temperature values. **`UTEMP`** will be called for each node given on the data lines.

If values are also given on the data lines, these values will be ignored. If a results or an output database file has been specified in addition to user subroutine **`UTEMP`**, values read from this file will be passed into **`UTEMP`** for possible modification.

Data lines to define gradients of temperature in beams and shells:

First line:

1. Node set or node number.
2. Reference temperature value. If the AMPLITUDE parameter is present, this value and subsequent gradient values will be modified by the AMPLITUDE specification.

3. Temperature gradient in the n_2-direction for beams or temperature gradient through the thickness for shells.
4. Temperature gradient in the n_1-direction for beams.

Repeat this data line as often as necessary to define temperatures at different nodes or node sets.

Data lines to define temperatures at temperature points in beams and shells:

First line:

1. Node set or node number.
2. Temperature at the first temperature point. If the AMPLITUDE parameter is present, this value and subsequent temperature values will be modified by the AMPLITUDE specification.
3. Temperature of the second temperature point.
4. Temperature of the third temperature point.
5. Etc., up to seven temperatures.

Subsequent lines (only needed if there are more than seven temperature points in the element):

1. Temperature of the eighth temperature point.
2. Etc., up to eight temperatures per line.

If more than seven temperature values are needed at any node, continue on the next line. It may be necessary to leave blank data lines for some nodes if any other node in the model has more than seven temperature points because the total number of temperatures that Abaqus expects to read for any node is based on the maximum number of temperature values of all the nodes in the model. These trailing initial values will be zero and will not be used in the analysis.

Repeat this set of data lines as often as necessary to define temperatures at different nodes or node sets.

Data lines to define temperatures for solid or frame elements using the data line format:

First line:

1. Node set or node number.
2. Temperature. If the AMPLITUDE parameter is present, this value will be modified by the AMPLITUDE specification.

Repeat this data line as often as necessary to prescribe temperature at different nodes or node sets.

No data lines are needed if temperatures are read from an Abaqus results or output database file and a user subroutine is not used (FILE parameter included, USER parameter omitted).

Data lines to define temperatures using user subroutine `UTEMP` using the data line format (FILE parameter omitted, USER parameter included):

First line:

1. Node set or node number.

Repeat this data line as often as necessary. **`UTEMP`** will be called for each node listed.

Data lines to define temperatures using user subroutine `UTEMP` when temperatures are read from an Abaqus results or output database file (FILE and USER parameters included):

First line:

1. Node set or node number.

Repeat this data line as often as necessary. The nodes identified on the data lines will be assigned values from the results or output database file; optionally, these values can be modified in user subroutine **`UTEMP`**.

19.2 *TENSILE FAILURE: Specify a tensile failure model and criterion.

This option is used with the Mises or the Johnson-Cook plasticity models or the equation of state model to specify a tensile failure model and criterion. It must be used in conjunction with the *PLASTIC, HARDENING=ISOTROPIC option; the *PLASTIC, HARDENING=JOHNSON COOK option; or the *EOS option.

Product: Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Equation of state," Section 22.2.1 of the Abaqus Analysis User's Manual
- "Classical metal plasticity," Section 20.2.1 of the Abaqus Analysis User's Manual
- "Johnson-Cook plasticity," Section 20.2.7 of the Abaqus Analysis User's Manual
- "Dynamic failure models," Section 20.2.8 of the Abaqus Analysis User's Manual
- *EOS
- *PLASTIC

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the hydrostatic cutoff stress, in addition to temperature. If this parameter is omitted, it is assumed that the hydrostatic cutoff stress is constant or depends only on temperature. See "Using the DEPENDENCIES parameter to define field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

ELEMENT DELETION

Set ELEMENT DELETION=YES (default) to allow element deletion when the failure criterion is met.

Set ELEMENT DELETION=NO to allow BRITTLE/DUCTILE-type failure for the deviatoric and hydrostatic parts of stresses.

Required parameters for use with ELEMENT DELETION=NO:

PRESSURE

Set PRESSURE=BRITTLE to model the case where the pressure stress is required to be compressive when the failure criterion is met.

Set PRESSURE=DUCTILE to model the case where the pressure stress will be limited by the hydrostatic cutoff stress when the failure criterion is met.

SHEAR

Set SHEAR=BRITTLE to model the case where the deviatoric stresses will be set to zero when the failure criterion is met.

Set SHEAR=DUCTILE to model the case where the deviatoric stresses will be unaffected when the failure criterion is met.

Data lines to specify a tensile failure model:

First line:

1. Hydrostatic cutoff stress (positive in tension). (Units of FL^{-2}.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of hydrostatic cutoff stress as a function of temperature and other predefined field variables.

19.3 *TENSION CUTOFF: Specify tension cutoff data for the Mohr-Coulomb plasticity model.

This option is used to specify tension cutoff data to limit the load carrying capacity of the Mohr-Coulomb plasticity model near the tensile region. This option must be used in conjunction with *MOHR COULOMB and *MOHR COULOMB HARDENING options.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Mohr-Coulomb plasticity," Section 20.3.3 of the Abaqus Analysis User's Manual
- *MOHR COULOMB
- *MOHR COULOMB HARDENING

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the tension cutoff stress, in addition to temperature. If this parameter is omitted, it is assumed that the tensile yield stress depends only on the plastic strain and, possibly, on temperature.

Data lines to define tension cutoff:

First data line:

1. Yield stress in uniaxial tension, σ_t.
2. Corresponding plastic strain. (The first tabular value entered must always be zero.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent data lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of tension cutoff on predefined field variables.

19.4 *TENSION STIFFENING: Define the retained tensile stress normal to a crack in a *CONCRETE model.

This option is used to define the retained tensile stress normal to a crack as a function of the deformation in the direction of the normal to the crack. It must be used with and appear after the *CONCRETE option. The *TENSION STIFFENING option can also be used in conjunction with the *SHEAR RETENTION and *FAILURE RATIOS options.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Concrete smeared cracking," Section 20.6.1 of the Abaqus Analysis User's Manual
- *CONCRETE
- *FAILURE RATIOS
- *SHEAR RETENTION

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the postcracking behavior, in addition to temperature. If this parameter is omitted, it is assumed that the postcracking behavior depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

TYPE

Set TYPE=DISPLACEMENT to define the postcracking behavior by entering the displacement, u_0, at which a linear loss of strength after cracking gives zero stress.

Set TYPE=STRAIN (default) to specify the postcracking behavior by entering the postfailure stress-strain relationship directly.

Data lines if the parameter TYPE=STRAIN is included (default):

First line:

1. Fraction of remaining stress to stress at cracking.
2. Absolute value of the direct strain minus the direct strain at cracking.
3. Temperature.

4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

The first point at each value of temperature must be a stress fraction of 1.0 at a strain of 0.0.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

Data lines if the parameter TYPE=DISPLACEMENT is included:

First line:

1. Displacement, u_0, at which a linear loss of strength after cracking gives zero stress. (Units of L.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the postcracking behavior on temperature and other predefined field variables.

19.5 *THERMAL EXPANSION: Define the thermal expansion behavior of beams.

This option can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Using a general beam section to define the section behavior," Section 26.3.7 of the Abaqus Analysis User's Manual
- *BEAM GENERAL SECTION

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies for the thermal-expansion coefficient, in addition to temperature. If this parameter is omitted, it is assumed that the thermal-expansion coefficient is constant or depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define the thermal expansion behavior:

First line:

1. Coefficient of thermal expansion.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the thermal expansion coefficient as a function of temperature and other predefined field variables.

19.6 *TIE: Define surface-based tie and cyclic symmetry constraints or coupled acoustic-structural interactions.

This option is used to impose tie constraints, cyclic symmetry constraints, or coupled acoustic-structural interactions between pairs of surfaces.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Interaction module

References:

- "Mesh tie constraints," Section 31.3.1 of the Abaqus Analysis User's Manual
- "Element-based surface definition," Section 2.3.2 of the Abaqus Analysis User's Manual
- "Node-based surface definition," Section 2.3.3 of the Abaqus Analysis User's Manual
- "Analysis of models that exhibit cyclic symmetry," Section 10.4.3 of the Abaqus Analysis User's Manual
- "Acoustic, shock, and coupled acoustic-structural analysis," Section 6.10.1 of the Abaqus Analysis User's Manual

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the constraint.

Optional, mutually exclusive parameters:

POSITION TOLERANCE

Set this parameter equal to a cutoff distance that is used to determine which nodes on the slave surface are tied to the master surface. The calculation of the distance between the slave and master surface for a particular slave node depends on factors such as shell element thickness, the setting of the TYPE parameter, and the types of surfaces involved. Slave nodes that do not satisfy the position tolerance are not tied to the master surface. The default value for this tolerance distance depends on the type of formulation and surfaces used in the constraints.

TIED NSET

Set this parameter equal to the node set label of the node set that includes the nodes on the slave surface that will be tied to the master surface. Nodes not included in this node set will not be tied.

Optional parameters:

ADJUST

Set ADJUST=YES (default) to move all tied nodes on the slave surface onto the master surface in the initial configuration, without any strain.

Set ADJUST=NO if the slave nodes will not be moved. This is the default if the slave surface belongs to a substructure or if one or more of the surfaces is beam element-based.

CONSTRAINT RATIO

This parameter applies only when two surfaces with rotational degrees of freedom are tied with an offset but the NO ROTATION parameter is used.

Set this parameter equal to the fractional distance between the master reference surface and the slave node at which the translational constraint should act. By default, Abaqus will attempt to choose this distance such that the translational constraint acts precisely at the interface.

CYCLIC SYMMETRY

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to invoke a constraint between the faces bounding a repetitive sector of a cyclic symmetric structure. This parameter can be used only in conjunction with the *CYCLIC SYMMETRY MODEL option.

NO ROTATION

Include this parameter if rotation degrees of freedom should not be tied. If this parameter is omitted, any existing rotation degrees of freedom will be tied if applicable, in addition to the translation degrees of freedom.

NO THICKNESS

Include this parameter to ignore shell thickness effects in calculations involving position tolerances and adjustments for initial gaps.

TYPE

Set TYPE=SURFACE TO SURFACE (default for most cases in Abaqus/Standard) to have the tie coefficients generated such that stress accuracy is optimized for the specified surface type pairings.

Set TYPE=NODE TO SURFACE (default for all cases in Abaqus/Explicit) to have the tie coefficients generated according to the interpolation functions at the point where the slave node projects onto the master surface.

Data lines to define the surfaces forming the constraint pairs:

First line:

1. The slave surface name.
2. The master surface name.

Repeat this data line as often as necessary to define all the surfaces forming the constraint pairs. Each data line defines a pair of surfaces that will be tied together.

19.7 *TIME POINTS: Specify time points at which data are written to the output database file, or specify time points in the loading history at which the response of a structure will be evaluated in a direct cyclic analysis.

This option is used to specify time points at which data are written to the output database file or, if it is used in conjunction with the *DIRECT CYCLIC option, to specify time points in the loading history at which the response of a structure will be evaluated.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Step module

References:

- *DIRECT CYCLIC
- *OUTPUT

Required parameter:

NAME

Set this parameter equal to a label that will be used to refer to the time points.

Optional parameters:

GENERATE

If this parameter is included, each data line should give a starting time point, t_1; an ending time point, t_2; and the time increment between these two specified time points, Δt.

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

Data lines if the GENERATE parameter is omitted:

First line:

1. List of time points; the points must be arranged in ascending order.

Repeat this data line as often as necessary. Up to eight entries are allowed per line. If you use the *TIME POINTS option in conjunction with the *DIRECT CYCLIC option, the listed time points must include the starting time and ending time in a single loading cycle. The time points must be specified in the step time.

Data lines if the GENERATE parameter is included:

First line:

1. Starting time point.
2. Ending time point, which must be later than the starting time point.
3. Increment in time between the two specified time points.

Repeat this data line as often as necessary. If you use the *TIME POINTS option in conjunction with the *DIRECT CYCLIC option, the listed time points must include the starting time and ending time in a single loading cycle. The time points must be specified in the step time.

19.8 *TORQUE: Define the torsional behavior of beams.

This option can be used only in conjunction with the *BEAM GENERAL SECTION, SECTION=NONLINEAR GENERAL option.

Products: Abaqus/Standard Abaqus/Explicit

Type: Model data

Level: Part, Part instance

References:

- "Using a general beam section to define the section behavior," Section 26.3.7 of the Abaqus Analysis User's Manual
- *BEAM GENERAL SECTION

Optional parameters (if neither ELASTIC nor LINEAR is included, elastic-plastic response is assumed):

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the torque–twist relationship, in addition to temperature. If this parameter is omitted, it is assumed that the torque–twist relationship is constant or depends only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

ELASTIC

Include this parameter if the torque–twist relationship is nonlinear but elastic.

LINEAR

Include this parameter if the torque varies linearly with twist.

Data lines if the LINEAR parameter is included:

First line:

1. Torsional stiffness of the section.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the torsional stiffness as a function of temperature and other predefined field variables.

Data lines if the LINEAR parameter is omitted:

First line:

1. Torque.
2. Twist.
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the torsional behavior as a function of temperature and other predefined field variables.

19.9 *TORQUE PRINT: Print a summary of the total torque that can be transmitted across axisymmetric slide lines.

This option is used to obtain a summary of the total torque that can be transmitted across all axisymmetric slide lines in a model.

Product: Abaqus/Standard

Type: History data

Level: Step

Reference:

- "Slide line contact elements," Section 36.4.1 of the Abaqus Analysis User's Manual

Optional parameter:

FREQUENCY

Set this parameter equal to the output frequency, in increments. When this option is used, the torque output will always be printed at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1.

There are no data lines associated with this option.

19.10 *TRACER PARTICLE: Define tracer particles for tracking the location of and results at material points during a step.

This option is used to define tracer particles and assign them to tracer sets for tracking the location of and results at material points during a step. The tracer set name is used in conjunction with the *ELEMENT OUTPUT and/or the *NODE OUTPUT options to request output for the tracer particles associated with the tracer set name.

Product: Abaqus/Explicit

Type: History data

Level: Step

References:

- "Output to the output database," Section 4.1.3 of the Abaqus Analysis User's Manual
- *ELEMENT OUTPUT
- *NODE OUTPUT

Required parameter:

TRACER SET

Set this parameter equal to the name of the tracer set to which these tracer particles will be assigned.

Optional parameter:

PARTICLE BIRTH STAGES

Set this parameter equal to the number of tracer particle births within the step. If this parameter is omitted, a single particle birth will occur at the beginning of the step. If this parameter has a value n greater than one, tracer particles will leave their parent nodes n times during the step at equally spaced intervals in time.

Data lines to define the tracer particles associated with the tracer set:

First line:

1. List of nodes or node set labels to be assigned as tracer particles corresponding to this tracer set.

Repeat this data line as often as necessary. Up to 16 entries are allowed per line.

19.11 *TRANSFORM: Specify a local coordinate system at nodes.

This option is used to specify a local coordinate system for displacement and rotation degrees of freedom at a node.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Assembly

Abaqus/CAE: Define nodal coordinate systems for prescribed conditions in the Load module.

Reference:

- "Transformed coordinate systems," Section 2.1.5 of the Abaqus Analysis User's Manual

Required parameter:

NSET

Set this parameter equal to the name of the node set for which the local transformed system is being given.

Optional parameter:

TYPE

Set TYPE=R (default) to indicate a rectangular Cartesian system (Figure 19.11–1). Set TYPE=C to indicate a cylindrical system (Figure 19.11–2). Set TYPE=S to indicate a spherical system (Figure 19.11–3).

Data line to define a transformed coordinate system:

First (and only) line:

1. Global X-coordinate of point a specifying transformation.
2. Global Y-coordinate of point a specifying transformation.
3. Global Z-coordinate of point a specifying transformation.
4. Global X-coordinate of point b specifying transformation.
5. Global Y-coordinate of point b specifying transformation.
6. Global Z-coordinate of point b specifying transformation.

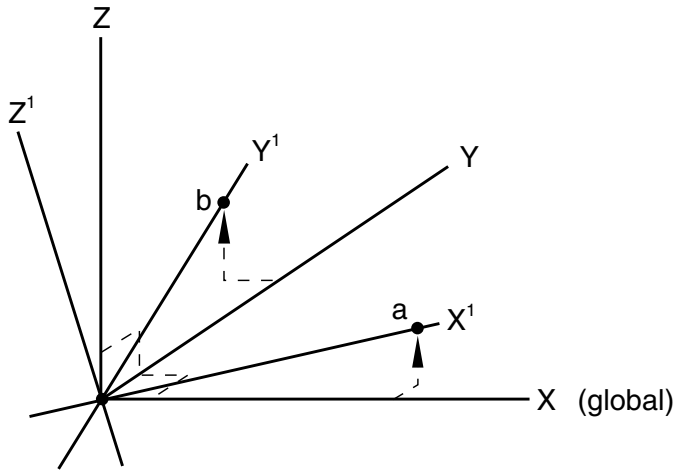


Figure 19.11–1 Cartesian transformation option.

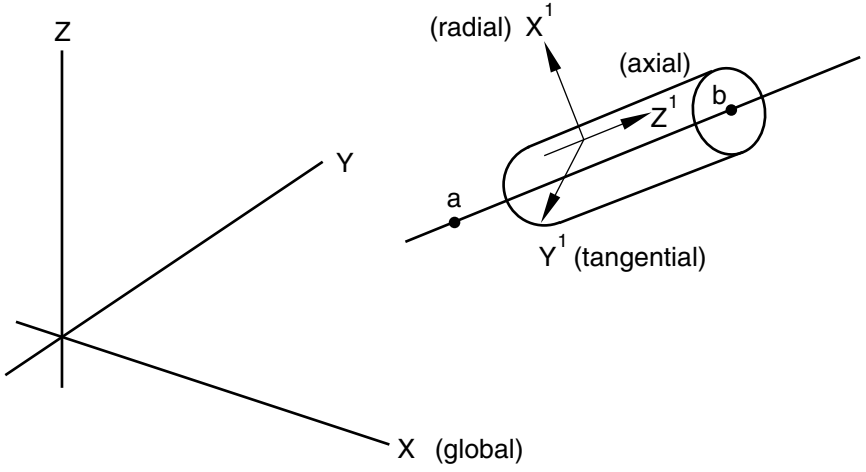


Figure 19.11–2 Cylindrical transformation option.

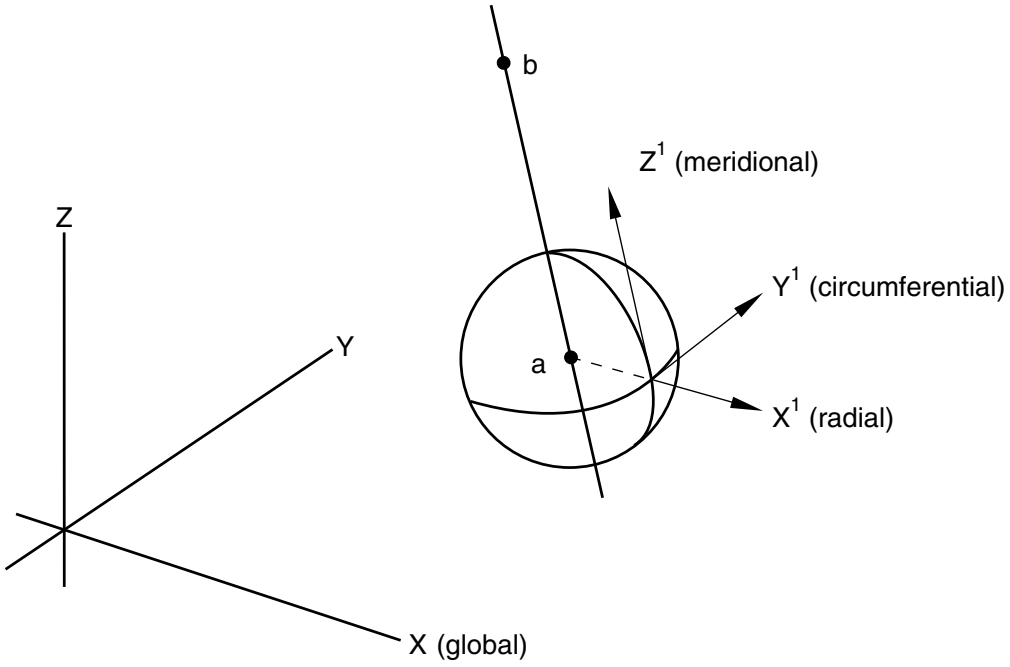


Figure 19.11–3 Spherical transformation option.

19.12 *TRANSPORT VELOCITY: Specify angular transport velocity.

This option is used to define the angular velocity of material transported through the mesh of a deformable body or the transport of material relative to the reference node of a rigid body during a steady-state transport analysis.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Steady-state transport analysis," Section 6.4.1 of the Abaqus Analysis User's Manual
- "Symmetric model generation," Section 10.4.1 of the Abaqus Analysis User's Manual
- "UMOTION," Section 1.1.39 of the Abaqus User Subroutines Reference Manual

Optional parameters:

AMPLITUDE

Set this parameter equal to the name of the amplitude curve (defined in the *AMPLITUDE option) that gives the time variation of the velocity throughout the step ("Amplitude curves," Section 30.1.2 of the Abaqus Analysis User's Manual).

If this parameter is omitted, the reference magnitude is applied immediately at the beginning of the step or linearly over the step, depending on the value assigned to the AMPLITUDE parameter on the *STEP option (see "Procedures: overview," Section 6.1.1 of the Abaqus Analysis User's Manual).

USER

Include this parameter to indicate that the magnitude of the rotational velocity will be defined in user subroutine **UMOTION**. If this parameter is used, any magnitudes defined by the data lines can be redefined in the user subroutine.

Data lines to define rotational motion:

First line:

1. Node set label or node number.
2. Magnitude of the rotational velocity (in radians/time). This magnitude will be modified by the *AMPLITUDE specification if the AMPLITUDE parameter is used. For a deformable body the rotation is assumed to be about the axis defined on the *SYMMETRIC MODEL

GENERATION option. For a rigid body of type REVOLUTION the rotation is assumed to be about the axis of revolution of the body.

Repeat this data line as often as necessary to define rotational motion on nodes of different parts of the model.

19.13 *TRANSVERSE SHEAR STIFFNESS: Define transverse shear stiffness for beams and shells.

This option must be used in conjunction with the *BEAM GENERAL SECTION option, the *BEAM SECTION option, the *COHESIVE SECTION option, the *SHELL GENERAL SECTION option, or the *SHELL SECTION option. The transverse shear stiffness defined with this option affects only the transverse shear flexible elements whose section properties are defined by the immediately preceding section option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance

Abaqus/CAE: Property module

References:

- "Shell section behavior," Section 26.6.4 of the Abaqus Analysis User's Manual
- "Choosing a beam element," Section 26.3.3 of the Abaqus Analysis User's Manual
- "Defining the constitutive response of cohesive elements using a continuum approach," Section 29.5.5 of the Abaqus Analysis User's Manual
- "Defining the constitutive response of cohesive elements using a traction-separation description," Section 29.5.6 of the Abaqus Analysis User's Manual
- *SHELL GENERAL SECTION
- *SHELL SECTION
- *BEAM GENERAL SECTION
- *BEAM SECTION
- *COHESIVE SECTION

There are no parameters associated with this option.

Data line when used with cohesive sections, shell sections, and *BEAM GENERAL SECTION, SECTION=MESHED:

First (and only) line:

1. Value of the shear stiffness of the section in the first direction, K_{11}^{ts}.
2. Value of the shear stiffness of the section in the second direction, K_{22}^{ts}.
3. Value of the coupling term in the shear stiffness of the section, K_{12}^{ts}.

If either value K_{11}^{ts} or K_{22}^{ts} is omitted or given as zero, the nonzero value will be used for both.

Data line when used with all other beam sections:

First (and only) line:

1. Value of the K_{23} shear stiffness of the section.
2. Value of the K_{13} shear stiffness of the section.
3. Value of the slenderness compensation factor or the label SCF. If this field is left blank, a default value of 0.25 is assumed. If the label SCF is specified, the values of the shear stiffness specified by the user will be ignored. They and the slenderness compensation factor will be calculated from the elastic material definition with the beam section.

If either value $K_{\alpha 3}$ is omitted or given as zero, the nonzero value will be used for both when the label SCF is not used.

19.14 *TRIAXIAL TEST DATA: Provide triaxial test data.

This option is required if some or all of the material parameters that define the exponent form of the *DRUCKER PRAGER option are to be calibrated from triaxial test data.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "Extended Drucker-Prager models," Section 20.3.1 of the Abaqus Analysis User's Manual

Optional parameters:

A

Set this parameter equal to the value of the material constant a if it is known and is held fixed at the input value. Omit the parameter and its value if a is to be obtained from the calibration.

B

Set this parameter equal to the value of the material constant b if it is known and is held fixed at the input value. Omit the parameter and its value if b is to be obtained from the calibration.

PT

Set this parameter equal to the value of the material constant p_t if it is known and is held fixed at the input value. Omit the parameter and its value if p_t is to be obtained from the calibration.

Data lines to specify triaxial test data:

First line:

1. Sign and magnitude of confining stress, $\sigma_1 = \sigma_2$.
2. Sign and magnitude of the stress in loading direction, σ_3.

Repeat this data line as often as necessary to give the yield stress at different levels of confining stress.

19.15 *TRS: Used to define temperature-time shift for time history viscoelastic analysis.

This option can be used only in conjunction with the *VISCOELASTIC option and, in Abaqus/Explicit, with the *VISCOSITY option.

In an Abaqus/Standard analysis, viscoelasticity must be defined in the time domain by using the *VISCOELASTIC option with the TIME parameter.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Time domain viscoelasticity," Section 19.7.1 of the Abaqus Analysis User's Manual
- "UTRS," Section 1.1.48 of the Abaqus User Subroutines Reference Manual
- "VUTRS," Section 1.2.19 of the Abaqus User Subroutines Reference Manual
- *VISCOELASTIC
- *VISCOSITY

Optional parameters:

DEFINITION

Set DEFINITION=WLF (default) to define the shift function by the Williams-Landell-Ferry approximation.

Set DEFINITION=ARRHENIUS to define the shift function by the Arrhenius approximation.

Set DEFINITION=USER to define the shift function in user subroutine **UTRS** in Abaqus/Standard analyses or in user subroutine **VUTRS** in Abaqus/Explicit analyses.

PROPERTIES

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of properties being entered. The properties are available for use in user subroutine **VUTRS**.

Data line to define the shift function by the Williams-Landell-Ferry approximation:

First (and only) line:

1. Reference temperature, θ_0.
2. Calibration constant, C_1.
3. Calibration constant, C_2.

Data line to define the shift function by the Arrhenius approximation:

First (and only) line:

1. Reference temperature, θ_0.
2. Activation energy, E_0.

In addition, you need to specify the universal gas constant and absolute zero using the *PHYSICAL CONSTANTS option.

To define a user-defined shift function:

No data lines are used with this option in Abaqus/Standard analyses when DEFINITION=USER is specified. Instead, user subroutine **UTRS** must be used to define the shift function.

For Abaqus/Explicit analyses in which DEFINITION=USER is specified, data lines are used if the PROPERTIES parameter is also specified. In all cases user subroutine **VUTRS** must be used to define the shift function. The properties, if entered, are available for use in user subroutine **VUTRS**.

Data lines to define material properties when PROPERTIES is specified:

First line:

1. Enter the material properties, eight per line.

Repeat this data line as often as necessary to define all material properties.

20. U

20.1 *UEL PROPERTY: Define property values to be used with a user element type.

This option is used to define the properties of a user element.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Model

Abaqus/CAE: Actuator/sensor interaction properties can be defined in the Interaction module.

References:

- "User-defined elements," Section 29.16.1 of the Abaqus Analysis User's Manual
- *USER ELEMENT

Required parameter:

ELSET

Set this parameter equal to the name of the element set containing the user elements for which these property values are being defined.

Optional parameters:

MATERIAL

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of the material to be used with these elements.

ORIENTATION

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the name of an orientation definition ("Orientations," Section 2.2.5 of the Abaqus Analysis User's Manual) to be used to define a local coordinate system for material calculations in the elements in this set.

Optional parameters (relevant only for direct-integration dynamic analysis with linear user elements):

ALPHA

Set this parameter equal to the Rayleigh mass damping factor, α.

BETA

Set this parameter equal to the Rayleigh stiffness damping factor, β.

To define the properties of linear user elements:

There are no data lines required.

Data lines to define the properties of nonlinear user elements if the PROPERTIES and/or I PROPERTIES parameters are used on the *USER ELEMENT option with a value of one or more:

First line:

1. Enter the values of the element properties. Enter all floating point values first, followed immediately by the integer values.

Repeat this data line as often as necessary. Eight values per line are used for both real and integer values.

20.2 *UNDEX CHARGE PROPERTY: Define an UNDEX charge for incident waves.

This option defines parameters that create the time histories of load, displacement, and other variables used to simulate an underwater explosion. This option must be used in conjunction with the *INCIDENT WAVE INTERACTION PROPERTY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model or history data

Level: Model, Step

Abaqus/CAE: Interaction module

References:

- "Acoustic and shock loads," Section 30.4.5 of the Abaqus Analysis User's Manual
- *INCIDENT WAVE INTERACTION PROPERTY

There are no parameters associated with this option.

Data lines to define the UNDEX charge:

First line:

1. Charge material constant, K.
2. Charge material constant, k.
3. Charge material constant, A.
4. Charge material constant, B.
5. Adiabatic charge constant, K_c.
6. Ratio of specific heats for gas, γ.

Second line:

1. Density of charge material, ρ_c.
2. Mass of charge material, m_c.
3. Acceleration due to gravity, g.
4. Atmospheric pressure, p_{atm}.
5. Wave effect parameter, η. Set to 1.0 for wave effects in the fluid and gas; set to 0.0 to neglect these effects. The default is 1.0.
6. Flow drag coefficient, C_D. The default is 0.0.
7. Flow drag exponent, E_D ($E_D \geq 0$). The default is 2.0.

Third line:

1. Time duration, T_{final}.
2. Maximum number of time steps for the bubble simulation, N_{steps}. The bubble amplitude simulation ceases when the number of steps reaches N_{steps} or the time duration, T_{final}, is reached. The default is 1500.
3. Relative step size control parameter, Ω_{rel}. The default is 1×10^{-11}.
4. Absolute step size control parameter, X_{abs}. The default is 1×10^{-11}.
5. Step size control exponent, β. The step size, Δt, is decreased or increased according to the error estimate: $(\Omega_{\text{rel}}|x| + X_{\text{abs}})^{\beta} \leq \Delta t \left|\frac{dx}{dt}\right|^{\beta}$. The default is 0.2.

Fourth line:

1. Depth magnitude of charge material, d_I.
2. *X*-direction cosine of fluid surface normal.
3. *Y*-direction cosine of fluid surface normal.
4. *Z*-direction cosine of fluid surface normal.

20.3 *UNIAXIAL: Characterize fabric materials through loading and unloading test data.

This option is used to indicate the start of shear or uniaxial test data along a particular direction to define the behavior of a fabric material. It must be used in conjunction with the *FABRIC option.

Product: Abaqus/Explicit

Type: Model data

Level: Model

References:

- "Fabric material behavior," Section 20.4.1 of the Abaqus Analysis User's Manual
- *FABRIC
- *LOADING DATA
- *UNLOADING DATA

Required parameter:

COMPONENT

Set COMPONENT=1 to define uniaxial behavior of fabric fibers in the "fill" direction.
Set COMPONENT=2 to define uniaxial behavior of fabric fibers in the "warp" direction.
Set COMPONENT=SHEAR to define shear response of the fabric.

There are no data lines associated with this option.

20.4 *UNIAXIAL TEST DATA: Used to provide uniaxial test data (compression and/or tension).

This option is used to provide uniaxial test data. It can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, the *LOW DENSITY FOAM option, and the *MULLINS EFFECT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Using uniaxial test data to define a hyperelastic material

References:

- "Hyperelastic behavior of rubberlike materials," Section 19.5.1 of the Abaqus Analysis User's Manual
- *HYPERELASTIC

Optional parameter:

SMOOTH

Include this parameter to apply a smoothing filter to the stress-strain data. If the parameter is omitted, no smoothing is performed.

Set this parameter equal to the number n such that $2n + 1$ is equal to the total number of data points in the moving window through which a cubic polynomial is fit using the least-squares method. n should be larger than 1. The default is SMOOTH=3.

Optional parameter when the *UNIAXIAL TEST DATA option is used in conjunction with the *HYPERELASTIC, MARLOW option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature.

Data lines to specify uniaxial test data for hyperelasticity other than the Marlow model (the nominal strains must be arranged in either ascending or descending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_U.

2. Nominal strain, ϵ_U.

Repeat this data line as often as necessary to give the stress-strain data.

Data lines to specify uniaxial test data for the Marlow model (the nominal strains must be arranged in ascending order if the SMOOTH parameter is used):

First line:

1. Nominal stress, T_U.
2. Nominal strain, ϵ_U.
3. Nominal lateral strain, $\epsilon_2 = \epsilon_3$. Not needed if the POISSON parameter is specified on the *HYPERELASTIC option or if the *VOLUMETRIC TEST DATA option is used.
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. Nominal strains and nominal stresses must be given in ascending order.

Using uniaxial test data to define an elastomeric foam

References:

- "Hyperelastic behavior in elastomeric foams," Section 19.5.2 of the Abaqus Analysis User's Manual
- *HYPERFOAM

There are no parameters associated with this option.

Data lines to specify uniaxial test data for a hyperfoam:

First line:

1. Nominal stress, T_L.
2. Nominal strain, ϵ_U.
3. Nominal lateral strain, $\epsilon_2 = \epsilon_3$. Default is zero. Not needed if the POISSON parameter is specified on the *HYPERFOAM option.

Repeat this data line as often as necessary to give the stress-strain data.

Using uniaxial test data to define a low-density foam material

References:

- "Low-density foams," Section 19.9.1 of the Abaqus Analysis User's Manual
- *LOW DENSITY FOAM

Required parameter:

DIRECTION

Set DIRECTION=TENSION to define tensile behavior.
Set DIRECTION=COMPRESSION to define compressive behavior.

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature.

Data lines to specify uniaxial test data for a low-density foam:

First line:

1. Nominal stress, T_U. (Provide the absolute value if DIRECTION=COMPRESSION.)
2. Nominal strain, ϵ_U. (Provide the absolute value if DIRECTION=COMPRESSION.)
3. Nominal strain rate, $\dot{\epsilon}_U$. (Provide positive values to specify the loading response and negative values to specify unloading.)
4. Temperature, θ.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. Nominal strains, nominal strain rates and nominal stresses must be given in ascending order.

Using uniaxial test data to define the Mullins effect material model

References:

- "Mullins effect in rubberlike materials," Section 19.6.1 of the Abaqus Analysis User's Manual
- "Energy dissipation in elastomeric foams," Section 19.6.2 of the Abaqus Analysis User's Manual
- *MULLINS EFFECT

There are no parameters associated with this option.

Data lines to specify uniaxial test data for defining the unloading-reloading response of the Mullins effect material model:

First line:

1. Nominal stress, T_L.
2. Nominal strain, ϵ_U.

Repeat this data line as often as necessary to give the stress-strain data.

20.5 *UNLOADING DATA: Provide unloading data for uniaxial behavior models in connectors or unloading data from uniaxial and shear tests for fabrics.

This option is used to define unloading response for the uniaxial behavior of connector elements when used in conjunction with the *CONNECTOR BEHAVIOR, *CONNECTOR UNIAXIAL BEHAVIOR, and *LOADING DATA options.

This option is used to define the unloading response from a uniaxial or a shear test for fabric materials when used in conjunction with the *FABRIC, *UNIAXIAL, and *LOADING DATA options. A fabric uniaxial test is specified with increasing strains along the specified yarn direction. A fabric shear test is specified with increasing shear strains as the fill and the warp yarns rotate with respect to each other.

Product: Abaqus/Explicit

Type: Model data

Level: Model

Defining the unloading response for uniaxial behavior in connectors

References:

- "Connector uniaxial behavior," Section 28.2.10 of the Abaqus Analysis User's Manual
- *CONNECTOR BEHAVIOR
- *CONNECTOR UNIAXIAL BEHAVIOR
- *LOADING DATA

Required parameter:

DEFINITION

Set DEFINITION=COMBINED to define an unloading path based on the specified unloading curve and a transition slope to transition from the loading to the unloading curve.

Set DEFINITION=EXPONENTIAL to define an exponential unloading path.

Set DEFINITION=INTERPOLATED CURVE to define an unloading path based on an interpolation between the specified unloading curves.

Set DEFINITION=QUADRATIC to define a quadratic unloading path.

Set DEFINITION=SHIFTED CURVE to define an unloading path based on shifting the specified unloading curve to the point of unloading.

Optional parameter:

RATE DEPENDENT

Include this parameter to define rate-dependent unloading data. If this parameter is omitted, the data are assumed to be rate independent. This parameter can be used only if the loading data are elastic and rate dependent.

Data lines for DEFINITION=COMBINED to define rate-independent unloading behavior that does not depend on independent components:

First line:

1. Transition slope.

Second line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=COMBINED to define rate-independent unloading behavior that depends on independent components:

First line:

1. Transition slope.

Second line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Constitutive relative motion in the first independent component.
4. Constitutive relative motion in the second independent component.
5. Etc., up to N_i entries as identified on the loading data definition.
6. Temperature.

7. First field variable.
8. Second field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the unloading curve data.

Data line for DEFINITION=EXPONENTIAL and DEFINITION=QUADRATIC:

First (and only) line:

1. Energy dissipation factor.
2. Permanent deformation factor. (Should be defined only in conjunction with the *LOADING DATA, TYPE=PERMANENT DEFORMATION option.)

Data lines for DEFINITION=INTERPOLATED CURVE and DEFINITION=SHIFTED CURVE to define rate-independent unloading behavior (the RATE DEPENDENT parameter is omitted) that does not depend on independent components:

First line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=INTERPOLATED CURVE and DEFINITION=SHIFTED CURVE to define rate-independent unloading behavior (the RATE DEPENDENT parameter is omitted) that depends on independent components:

First line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Constitutive relative motion in the first independent component.
4. Constitutive relative motion in the second independent component.
5. Etc., up to N_i entries as identified on the loading data definition.

6. Temperature.
7. First field variable.
8. Second field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=INTERPOLATED CURVE to define rate-dependent unloading behavior (the RATE DEPENDENT parameter is included) that does not depend on independent components:

First line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Relative velocity. (Provide the absolute value.)
4. Temperature.
5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=INTERPOLATED CURVE to define rate-dependent unloading behavior (the RATE DEPENDENT parameter is included) that depends on independent components:

First line:

1. Force or moment. (Provide the absolute value.)
2. Constitutive relative displacement or rotation. (Provide the absolute value.)
3. Relative velocity. (Provide the absolute value.)
4. Connector relative position or constitutive relative motion in the first independent component.
5. Connector relative position or constitutive relative motion in the second independent component.
6. Etc., up to N_i entries as identified on the loading data definition.
7. Temperature.
8. First field variable.

If the number of data entries exceeds the limit of eight entries per line, continue the input on the next data line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Defining the unloading response data from uniaxial tests of fabric materials

References:

- "Fabric material behavior," Section 20.4.1 of the Abaqus Analysis User's Manual
- *FABRIC
- *UNIAXIAL
- *LOADING DATA

Required parameter:

DEFINITION

Set DEFINITION=COMBINED to define an unloading path based on the specified unloading curve and a transition slope to transition from the loading to the unloading curve.

Set DEFINITION=EXPONENTIAL to define an exponential unloading path.

Set DEFINITION=INTERPOLATED CURVE to define an unloading path based on an interpolation between the specified unloading curves.

Set DEFINITION=QUADRATIC to define a quadratic unloading path.

Set DEFINITION=SHIFTED CURVE to define an unloading path based on shifting the specified unloading curve to the point of unloading.

The available unloading path types depend on the behavior type assigned to the test data (as defined on the *LOADING DATA option).

Optional parameter:

RATE DEPENDENT

Include this parameter equal to define rate-dependent unloading data. If this parameter is omitted, the data are assumed to be rate independent. This parameter can be used only if the loading data are elastic and rate dependent.

Data lines for DEFINITION=COMBINED:

First line:

1. Transition slope.

Second line:

1. Nominal stress. (Provide the absolute value.)
2. Nominal strain. (Provide the absolute value.)

3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Do not repeat the first data line. Repeat the subsequent data lines as often as necessary to define the unloading curve data.

Data line for DEFINITION=EXPONENTIAL and DEFINITION=QUADRATIC:

First (and only) line:

1. Energy dissipation factor.
2. Permanent deformation factor. (Should be defined only in conjunction with the *LOADING DATA, TYPE=PERMANENT DEFORMATION option.)

Data lines for DEFINITION=INTERPOLATED CURVE and DEFINITION=SHIFTED CURVE to define uniaxial rate-independent unloading behavior (the RATE DEPENDENT parameter is omitted):

First line:

1. Nominal stress. (Provide the absolute value.)
2. Nominal strain. (Provide the absolute value.)
3. Temperature.
4. First field variable.
5. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

Data lines for DEFINITION=INTERPOLATED CURVE to define uniaxial rate-dependent unloading behavior (the RATE DEPENDENT parameter is included):

First line:

1. Nominal stress. (Provide the absolute value.)
2. Nominal strain. (Provide the absolute value.)
3. Strain rate. (Provide the absolute value.)
4. Temperature.

5. First field variable.
6. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the unloading curve data.

20.6 *USER DEFINED FIELD: Redefine field variables at a material point.

This material option is used to allow the values of field variables at a material point to be redefined within an increment via user subroutine **USDFLD** in Abaqus/Standard analyses or user subroutine **VUSDFLD** in Abaqus/Explicit analyses. If the *USER DEFINED FIELD option is used, it must appear within a *MATERIAL definition ("Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "USDFLD," Section 1.1.45 of the Abaqus User Subroutines Reference Manual
- "VUSDFLD," Section 1.2.18 of the Abaqus User Subroutines Reference Manual

Optional parameter:

PROPERTIES

This parameter applies only to Abaqus/Explicit analyses.

Set this parameter equal to the number of properties being entered. The properties are available for use in user subroutine **VUSDFLD**.

Data lines to define material properties when PROPERTIES is specified:

First line:

1. Enter the material properties, eight per line.

Repeat this data line as often as necessary to define all material properties.

20.7 *USER ELEMENT: Introduce a user-defined element type.

This option is used to introduce a linear or a general user-defined element. It must precede any reference to this user element on an *ELEMENT option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Part, Part instance, Model

Abaqus/CAE: Actuator/sensor interactions can be defined in the Interaction module.

Introducing a linear user-defined element (Abaqus/Standard only)

References:

- "User-defined elements," Section 29.16.1 of the Abaqus Analysis User's Manual
- *ELEMENT
- *MATRIX
- *UEL PROPERTY

Required parameter:

TYPE

Set this parameter equal to the element type key used to identify this element on the *ELEMENT option. The format of this type key must be U*n* in Abaqus/Standard, where *n* is a positive integer less than 10000. To use this element type, set TYPE=U*n* on the *ELEMENT option.

Optional parameters:

FILE

Set this parameter equal to the name of the results file (with no extension) from which the data are to be read. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names.

This parameter can be used only if the user-defined element type is linear and its stiffness and/or mass matrices are to be read from the Abaqus/Standard results file of a previous analysis (in which they were written by using the *ELEMENT MATRIX OUTPUT or *SUBSTRUCTURE MATRIX OUTPUT options). When this parameter is used, all values are taken from the results file.

For example, if the stiffness or mass being read from the results file is not symmetric, the UNSYMM parameter will be invoked automatically.

If this parameter is omitted, the data will be read from a standard input file.

INTEGRATION

This parameter applies only to Abaqus/Standard analyses.

Set this parameter equal to the number of integration points to be used in Gauss integration. This parameter must be used in conjunction with the TENSOR parameter.

TENSOR

This parameter applies only to Abaqus/Standard analyses.

Include this parameter to specify the element type. This parameter must be used in conjunction with the INTEGRATION parameter.

Set TENSOR=THREED to specify that it is a three-dimensional element in a stress/displacement or heat transfer analysis.

Set TENSOR=TWOD to specify that it is a two-dimensional element in a heat transfer analysis.

Set TENSOR=PSTRAIN to specify that it is a plane strain element in a stress/displacement analysis.

Set TENSOR=PSTRESS to specify that it is a plane stress element in a stress/displacement analysis.

Required parameters if the FILE parameter is included:

OLD ELEMENT

Set this parameter equal to the element number that was assigned to the element whose matrices are being read. This parameter can also be set to a substructure identifier to read a substructure matrix from an Abaqus/Standard results file.

STEP

Set this parameter equal to the step number in which the element matrix was written. This parameter is not required if using a substructure whose matrix was output during its generation.

INCREMENT

Set this parameter equal to the increment number in which the element matrix was written. This parameter is not required if using a substructure whose matrix was output during its generation.

Required parameters if the FILE parameter is omitted:

LINEAR

Include this parameter to indicate that the behavior of the element type is linear and is defined by a stiffness matrix and/or a mass matrix. The *MATRIX option is required to define the element's behavior.

NODES

Set this parameter equal to the number of nodes associated with an element of this type.

Optional parameters if the FILE parameter is omitted:

COORDINATES

Abaqus/Standard assigns space to store the coordinate values at each node in user subroutine **UEL**. The default number of coordinate values is equal to the largest active degree of freedom of the user element with a maximum of 3. Use the COORDINATES parameter to increase the number of coordinate values.

UNSYMM

Include this parameter if the element matrices are not symmetric. This parameter will cause Abaqus/Standard to use its unsymmetric equation solution capability.

The presence or absence of this parameter determines the form in which the matrices must be provided for reading.

Data lines if the FILE parameter is omitted:

First line:

1. Enter the list of active degrees of freedom at the first node of the element (as determined by the connectivity list). The rule in "Conventions," Section 1.2.2 of the Abaqus Analysis User's Manual, regarding which degrees of freedom can be used for displacement, rotation, temperature, etc. must be conformed to.

Second line if the active degrees of freedom are different at subsequent nodes:

1. Enter the position in the connectivity list (node position on the element) where the new list of active degrees of freedom first applies.
2. Enter the new list of active degrees of freedom.

Repeat the second data line as often as necessary.

Introducing a general user-defined element

References:

- "User-defined elements," Section 29.16.1 of the Abaqus Analysis User's Manual
- "VUEL," Section 1.2.10 of the Abaqus User Subroutines Reference Manual
- "UEL," Section 1.1.23 of the Abaqus User Subroutines Reference Manual
- *ELEMENT
- *UEL PROPERTY

Required parameters:

TYPE

Set this parameter equal to the element type key used to identify this element on the *ELEMENT option. The format of this type key must be U*n* in Abaqus/Standard and VU*n* in Abaqus/Explicit, where *n* is a positive integer less than 10000. To use this element type, set TYPE=U*n* (or VU*n*) on the *ELEMENT option.

NODES

Set this parameter equal to the number of nodes associated with an element of this type.

Optional parameters:

COORDINATES

Set this parameter equal to the maximum number of coordinates needed in user subroutine **UEL** in Abaqus/Standard and user subroutine **VUEL** in Abaqus/Explicit at any node point of the element. Abaqus assigns space to store the coordinate values at all the nodes associated with elements of this type. The default is COORDINATES=1.

Abaqus will change the value of COORDINATES to be the maximum of the user-specified value of the COORDINATES parameter or the value of the largest active degree of freedom of the user element that is less than or equal to 3. For example, if COORDINATES=1 and the active degrees of freedom of the user element are 2, 3, and 6, the value of COORDINATES will be changed to 3. If COORDINATES=2 and the active degrees of freedom of the user element are 11 and 12, the value of COORDINATES will remain as 2.

I PROPERTIES

Set this parameter equal to the number of integer property values needed as data in user subroutine **UEL** (or **VUEL**) to define such an element. The default is I PROPERTIES=0.

PROPERTIES

Set this parameter equal to the number of real (floating point) property values needed as data in user subroutine **UEL** (or **VUEL**) to define such an element. The default is PROPERTIES=0.

UNSYMM

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the element matrices are not symmetric. This parameter will cause Abaqus/Standard to use its unsymmetric equation solution capability.

VARIABLES

Set this parameter equal to the number of solution-dependent state variables that must be stored within the element. Its value must be greater than 0. The default is VARIABLES=1.

Data lines to define a general user-defined element:

First line:

1. Enter the list of active degrees of freedom at the first node of the element (as determined by the connectivity list). The rule in "Conventions," Section 1.2.2 of the Abaqus Analysis User's Manual, regarding which degrees of freedom can be used for displacement, rotation, temperature, etc. must be conformed to.

Second line if the active degrees of freedom are different at subsequent nodes:

1. Enter the position in the connectivity list (node position on the element) where the new list of active degrees of freedom first applies.
2. Enter the new list of active degrees of freedom.

Repeat the second data line as often as necessary.

20.8 *USER MATERIAL: Define material constants for use in subroutine UMAT, UMATHT, or VUMAT.

This option is used to input material constants for use in a user-defined mechanical model (user subroutine **UMAT** in Abaqus/Standard or user subroutine **VUMAT** in Abaqus/Explicit). In Abaqus/Standard it is also used to input material constants for use in a user-defined thermal material model (user subroutine **UMATHT**).

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "User-defined mechanical material behavior," Section 23.8.1 of the Abaqus Analysis User's Manual
- "User-defined thermal material behavior," Section 23.8.2 of the Abaqus Analysis User's Manual
- "UMAT," Section 1.1.36 of the Abaqus User Subroutines Reference Manual
- "UMATHT," Section 1.1.37 of the Abaqus User Subroutines Reference Manual
- "VUMAT," Section 1.2.17 of the Abaqus User Subroutines Reference Manual

Required parameter:

CONSTANTS

Set this parameter equal to the number of constants being entered.

Optional parameters:

TYPE

This parameter applies only to Abaqus/Standard analyses.

Set TYPE=MECHANICAL (default) if the constants are used in defining the mechanical behavior of the material.

Set TYPE=THERMAL if the constants are used in defining the thermal constitutive behavior of the material.

If both the user-defined mechanical and the user-defined thermal behavior of the material are being modeled, the *USER MATERIAL option should be repeated in the material data block such that each value of the TYPE parameter is used.

UNSYMM

This parameter applies only to Abaqus/Standard analyses.

Include this parameter if the material stiffness matrix, $\partial \Delta \boldsymbol{\sigma} / \partial \Delta \boldsymbol{\varepsilon}$, is not symmetric or when a thermal constitutive model is used and $\partial \mathbf{f} / \partial (\partial \theta / \partial \mathbf{x})$ is not symmetric. This parameter causes Abaqus/Standard to use its unsymmetric equation solution procedures.

Data lines to define material constants:

First line:

1. Give the material constants, eight per line.

Repeat this data line as often as necessary to define all material constants.

20.9 *USER OUTPUT VARIABLES: Specify number of user variables.

This option is used to allow Abaqus to allocate space at each material calculation point for user-defined output variables defined in user subroutine **UVARM**. If the *USER OUTPUT VARIABLES option is used, it must appear within each of the relevant material or gasket behavior definitions.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Reference:

- "UVARM," Section 1.1.49 of the Abaqus User Subroutines Reference Manual

There are no parameters associated with this option.

Data line to specify the number of user variables:

First (and only) line:

1. Number of user-defined variables required at each material point.

21. V

21.1 *VARIABLE MASS SCALING: Specify mass scaling during the step.

This option is used to specify mass scaling during the step for part or all of the model.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Mass scaling," Section 11.7.1 of the Abaqus Analysis User's Manual
- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual

Optional parameters:

DT

Set this parameter equal to the desired element stable time increment for the element set provided. The mass scaling is applied according to the method specified with the TYPE parameter. If the DT parameter is omitted, all variable mass scaling definitions from previous steps are removed, and the scaled mass matrix from the end of the previous step is carried over to the current step.

ELSET

Set this parameter equal to the name of the element set for which this mass scaling definition is being applied. If this parameter is omitted, the mass scaling definition will apply to all elements in the model.

The *VARIABLE MASS SCALING option can be repeated with different ELSET definitions to define different mass scaling for the specified element sets.

TYPE

Set TYPE=UNIFORM to scale the masses of the elements equally so that the smallest element stable time increment of the scaled elements equals the value assigned to DT.

Set TYPE=BELOW MIN (default) to scale the masses of only the elements whose element stable time increments are less than the value assigned to DT. The masses of these elements will be scaled so that the element stable time increments equal the value assigned to DT.

Set TYPE=SET EQUAL DT to scale the masses of all elements so that they have the same element stable time increment equal to the value assigned to DT.

Set TYPE=ROLLING to scale the element masses automatically for the simulation of a rolling process. The appropriate value for the target stable time increment is determined by Abaqus from several parameters of the rolling process. The DT parameter will be ignored in this case.

Required, mutually exclusive parameters if the DT parameter or the TYPE=ROLLING parameter is used:

FREQUENCY

Set this parameter equal to the frequency, in increments, at which mass scaling calculations are to be performed during the step. For example, FREQUENCY=5 will scale the mass at the beginning of the step and at increments 5, 10, 15, etc. The value of this parameter must be a positive integer.

NUMBER INTERVAL

Set this parameter equal to the number of intervals during the step at which mass scaling calculations will be performed. For example, if NUMBER INTERVAL=2, mass scaling calculations will be performed at the beginning of the step, the increment immediately following the half-way point in the step, and the final increment in the step.

Required parameters for TYPE=ROLLING:

CROSS SECTION NODES

Set this parameter equal to the number of nodes in the cross-section of the workpiece. Increasing this value will decrease the amount of mass scaling used.

EXTRUDED LENGTH

Set this parameter equal to the average element length in the rolling direction.

FEED RATE

Set this parameter equal to the estimated average velocity of the workpiece in the rolling direction at steady-state conditions.

There are no data lines associated with this option.

21.2 *VIEWFACTOR OUTPUT: Write radiation viewfactors to the results file in cavity radiation heat transfer analysis.

This option is used to write cavity radiation element viewfactor matrices to the results file. This option is available only for heat transfer analysis including cavity radiation.

Product: Abaqus/Standard

Type: History data

Level: Step

References:

- "Cavity radiation," Section 37.1.1 of the Abaqus Analysis User's Manual
- "Output," Section 4.1.1 of the Abaqus Analysis User's Manual

Required parameter:

CAVITY

Set this parameter equal to the name of the cavity for which this output request is being made.

Optional parameter:

FREQUENCY

Set this parameter equal to the output frequency, in increments. The output will always be written at the last increment of each step unless FREQUENCY=0. The default is FREQUENCY=1. Set FREQUENCY=0 to suppress the output.

There are no data lines associated with this option.

21.3 *VISCO: Transient, static, stress/displacement analysis with time-dependent material response (creep, swelling, and viscoelasticity).

This option is used to obtain a transient static response in an analysis with time-dependent material behavior (creep, swelling, and viscoelasticity).

Products: Abaqus/Standard Abaqus/CAE

Type: History data

Level: Step

Abaqus/CAE: Step module

References:

- "Quasi-static analysis," Section 6.2.5 of the Abaqus Analysis User's Manual
- "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual

Optional parameters:

ALLSDTOL

Include this parameter to indicate that an adaptive automatic damping algorithm will be activated in this step. Set this parameter equal to the maximum allowable ratio of the stabilization energy to the total strain energy. The initial damping factor is specified via the STABILIZE parameter or the FACTOR parameter. This damping factor will then be adjusted through the step based on the convergence history and the value of ALLSDTOL. If this parameter is set equal to zero, the adaptive automatic damping algorithm is not activated; a constant damping factor will be used throughout the step. If this parameter is included without a specified value, the default value is 0.05. If this parameter is omitted but the STABILIZE parameter is included with the default value of dissipated energy fraction, the adaptive automatic damping algorithm will be activated automatically with ALLSDTOL=0.05.

This parameter must be used in conjunction with the STABILIZE parameter (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual).

CETOL

Set this parameter equal to the maximum difference in the creep strain increment calculated from the creep strain rates based on conditions at the beginning and on conditions at the end of the increment, thus controlling the accuracy of the creep integration. In general, the tolerance can be calculated by choosing an acceptable stress error tolerance and dividing by a typical elastic modulus. If the model consists of a linear viscoelastic material characterized by the *VISCOELASTIC option, a

looser tolerance of the order of the elastic strains can be used. If CETOL is omitted, fixed time increments are used.

CONTINUE

Set CONTINUE=NO (default) to specify that this step will not carry over the damping factors from the results of the preceding general step. In this case the initial damping factors will be recalculated based on the declared damping intensity and on the solution of the first increment of the step or can be specified directly.

Set CONTINUE=YES to specify that this step will carry over the damping factors from the end of the immediately preceding general step.

This parameter must be used in conjunction with the ALLSDTOL and the STABILIZE parameters.

CREEP

Set CREEP=EXPLICIT to use explicit integration for creep behavior throughout the step, which may sometimes be computationally less expensive. The time increment will be limited by the accuracy tolerance (CETOL) and by the stability limit of the forward difference operator. See "Rate-dependent plasticity: creep and swelling," Section 20.2.4 of the Abaqus Analysis User's Manual, for details on the integration scheme.

FACTOR

Set this parameter equal to the damping factor to be used in the automatic damping algorithm (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual) if the problem is expected to be unstable due to local instabilities and the damping factor calculated by Abaqus/Standard is not suitable. This parameter must be used in conjunction with the STABILIZE parameter and overrides the automatic calculation of the damping factor based on a value of the dissipated energy fraction.

STABILIZE

Include this parameter to use automatic stabilization if the problem is expected to be unstable due to local instabilities. Set this parameter equal to the dissipated energy fraction of the automatic damping algorithm (see "Solving nonlinear problems," Section 7.1.1 of the Abaqus Analysis User's Manual). If this parameter is omitted, the stabilization algorithm is not activated. If this parameter is included without a specified value, the default value of the dissipated energy fraction is 2×10^{-4} and the adaptive automatic damping algorithm will be activated by default with ALLSDTOL=0.05 in this step; set ALLSDTOL=0 to deactivate the adaptive automatic damping algorithm. If the FACTOR parameter is used, any value of the dissipated energy fraction will be overridden by the damping factor.

Data line for a transient, quasi-static analysis:

First (and only) line:

1. Suggested initial time increment. This value should be reasonably correct (in agreement with the strain change allowed with the CETOL parameter) but may be reduced if automatic time incrementation is being used.
2. Time period of the step.
3. Minimum time increment allowed. If Abaqus/Standard finds it needs a smaller time increment than this value, the analysis is terminated. If this entry is zero, a default value of the smaller of the suggested initial time increment or 10^{-5} times the total time period is assumed.
4. Maximum time increment allowed. Only used for automatic time incrementation. If this value is not specified, no upper limit is imposed.

21.4 *VISCOSITY: Specify material shear viscosity.

This option is used to specify the shear viscosity of the material. It must be used in conjunction with the *EOS option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Viscosity," Section 23.1.4 of the Abaqus Analysis User's Manual
- "Equation of state," Section 22.2.1 of the Abaqus Analysis User's Manual
- "VUVISCOSITY," Section 1.2.20 of the Abaqus User Subroutines Reference Manual
- *EOS
- *TRS

Optional parameters:

DEFINITION

Set DEFINITION=CARREAU-YASUDA to define the Carreau-Yasuda viscous shear behavior.

Set DEFINITION=CROSS to define the Cross viscous shear behavior.

Set DEFINITION=ELLIS-METER to define the Ellis-Meter viscous shear behavior.

Set DEFINITION=HERSCHEL-BULKEY to define the Herschel-Bulkey viscous shear behavior.

Set DEFINITION=NEWTONIAN (default) to define Newtonian (linear) viscous shear behavior.

Set DEFINITION=POWELL-EYRING to define the Powell-Eyring viscous shear behavior.

Set DEFINITION=POWER LAW to define the power law viscous shear behavior.

Set DEFINITION=TABULAR to define the non-Newtonian viscous shear behavior in tabular form.

Set DEFINITION=USER to define the viscous shear behavior in user subroutine **VUVISCOSITY**.

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the viscosity, in addition to temperature. If this parameter is omitted, it is assumed that the viscosity depends only on temperature. See "Using the DEPENDENCIES parameter to define field variable

dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

PROPERTIES

This parameter can be used only if DEFINITION=USER is specified.

Set this parameter equal to the number of property values needed as data in user subroutine **VUVISCOSITY**. The default value is 0.

Data lines to define the Carreau-Yasuda viscous shear behavior (DEFINITION=CARREAU-YASUDA):

First line:

1. Shear viscosity at low shear rates (Newtonian limit), η_0. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, η_∞. (Units of $FL^{-2}T$.)
3. Time constant, λ. (Units of T.)
4. Flow behavior index, n.
5. a.
6. Temperature.
7. First field variable.
8. Second field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than two):

1. Third field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Carreau-Yasuda viscosity model as a function of temperature and other predefined field variables.

Data lines to define the Cross viscous shear behavior (DEFINITION=CROSS):

First line:

1. Shear viscosity at low shear rates (Newtonian limit), η_0. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, η_∞. (Units of $FL^{-2}T$.)
3. Time constant, λ. (Units of T.)
4. Flow behavior index, n.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Cross viscosity model as a function of temperature and other predefined field variables.

Data lines to define the Ellis-Meter viscous shear behavior (DEFINITION=ELLIS-METER):

First line:

1. Shear viscosity at low shear rates (Newtonian limit), η_0. (Units of $\mathrm{FL^{-2}T}$.)
2. Shear viscosity at large shear rates, η_∞. (Units of $\mathrm{FL^{-2}T}$.)
3. Effective shear stress, $\tau_{1/2}$, at which the viscosity is 50% between the Newtonian limit, η_0, and the infinite shear viscosity, η_∞. (Units of $\mathrm{FL^{-2}}$.)
4. Flow behavior index, n.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Ellis-Meter viscosity model as a function of temperature and other predefined field variables.

Data lines to define the Herschel-Bulkey viscous shear behavior (DEFINITION=HERSCHEL-BULKEY):

First line:

1. Shear viscosity at low shear rates, η_0. (Units of $\mathrm{FL^{-2}T}$.)
2. Yield shear stress, τ_0. (Units of $\mathrm{FL^{-2}}$.)
3. Consistency index, k. (Units of $\mathrm{FL^{-2}T^{n}}$.)
4. Flow behavior index, n.
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Herschel-Bulkey viscosity model as a function of temperature and other predefined field variables.

Data lines to define Newtonian viscous shear behavior (DEFINITION=NEWTONIAN):

First line:

1. Viscosity. (Units of $FL^{-2}T$.)
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the viscosity as a function of temperature and other predefined field variables.

Data lines to define the Powell-Eyring viscous shear behavior (DEFINITION=POWELL-EYRING):

First line:

1. Shear viscosity at low shear rates (Newtonian limit), η_0. (Units of $FL^{-2}T$.)
2. Shear viscosity at large shear rates, η_∞. (Units of $FL^{-2}T$.)
3. Time constant, λ. (Units of T.)
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the Powell-Eyring viscosity model as a function of temperature and other predefined field variables.

Data lines to define the power law viscous shear behavior (DEFINITION=POWER LAW):

First line:

1. Consistency index, k. (Units of $\mathrm{FL^{-2}T}^n$.)
2. Flow behavior index, n.
3. Maximum viscosity, η_{max}. (Units of $\mathrm{FL^{-2}T}$.)
4. Minimum viscosity, η_{min}. (Units of $\mathrm{FL^{-2}T}$.)
5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the coefficients of the power law viscosity model as a function of temperature and other predefined field variables.

Data lines to define the viscous shear behavior in tabular form (DEFINITION=TABULAR):

First line:

1. Viscosity, η. (Units of $\mathrm{FL^{-2}T}$.)
2. Effective shear strain rate, $\dot{\gamma}$. (Units of $\mathrm{T^{-1}}$.)
3. Temperature.
4. First field variable.
5. Second field variable.
6. Etc., up to six field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than six):

1. Seventh field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the viscosity as a function of effective shear strain rate, temperature, and other predefined field variables.

Data lines to define the material properties for the user-defined viscosity model (DEFINITION=USER):

No data lines are needed if the PROPERTIES parameter is omitted or set to 0. Otherwise, first line:

1. Give the material properties, eight per line.

Repeat this data line as often as necessary to define the material properties.

21.5 *VISCOELASTIC: Specify dissipative behavior for use with elasticity.

This option is used to generalize a material's elastic response to include viscoelasticity. The viscoelasticity can be defined as a function of frequency for steady-state small-vibration analyses or as a function of reduced time for time-dependent analyses.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Time domain viscoelasticity," Section 19.7.1 of the Abaqus Analysis User's Manual
- "Frequency domain viscoelasticity," Section 19.7.2 of the Abaqus Analysis User's Manual
- *COMBINED TEST DATA
- *SHEAR TEST DATA
- *TRS
- *VOLUMETRIC TEST DATA

Required, mutually exclusive parameters:

FREQUENCY

This parameter applies only to Abaqus/Standard analyses.

Use this parameter to choose the frequency domain definition. In this case the material's long-term elasticity must be defined using the *ELASTIC or the *HYPERELASTIC option.

Set FREQUENCY=CREEP TEST DATA to define the frequency domain response using a Prony series representation of the relaxation moduli, if the Prony series parameters are to be computed from shear and volumetric creep test data.

Set FREQUENCY=FORMULA to define the dissipative material parameters by the power law formulæ.

Set FREQUENCY=PRONY to define the frequency domain response using a Prony series representation of the relaxation moduli by specifying the Prony series parameters.

Set FREQUENCY=RELAXATION TEST DATA to define the frequency domain response using a Prony series representation of the relaxation moduli, if the Prony series parameters are to be computed from shear and volumetric relaxation test data.

Set FREQUENCY=TABULAR to provide tabular definition of the frequency domain response.

TIME

Use this parameter to choose the time domain definition. In this case the material's elasticity must be defined using the *ELASTIC, the *HYPERELASTIC, or the *HYPERFOAM option.

Set TIME=CREEP TEST DATA if the Prony series parameters are to be computed by Abaqus from data taken from shear and volumetric creep tests.

Set TIME=FREQUENCY DATA if the Prony series parameters are to be computed by Abaqus from frequency-dependent cyclic test data.

Set TIME=PRONY to define a linear, isotropic, viscoelastic material by giving the parameters of the Prony series representation of the relaxation moduli.

Set TIME=RELAXATION TEST DATA if the Prony series parameters are to be computed by Abaqus from data taken from shear and volumetric relaxation tests.

Optional parameters:

PRELOAD

This parameter applies only to Abaqus/Standard analyses.

This parameter can be used only in conjunction with FREQUENCY=TABULAR to specify the nature of preload used for defining frequency-domain viscoelastic material properties or effective thickness-direction gasket properties.

Set PRELOAD=UNIAXIAL to specify that the frequency-domain viscoelastic material properties correspond to a uniaxial test.

Set PRELOAD=VOLUMETRIC to specify that the frequency-domain viscoelastic material properties correspond to a volumetric test. This setting is not meaningful when used with gasket elements to define effective thickness-direction properties.

TYPE

Use this parameter to define whether the *VISCOELASTIC option is being used to define continuum material properties or effective thickness-direction gasket properties.

Set TYPE=ISOTROPIC (default, and only option for Abaqus/Explicit) to define continuum material properties. This choice is appropriate when the viscoelasic material model is used for any continuum, structural, or special-purpose elements whose material response is modeled using continuum material properties (examples of such special-purpose elements include modeling cohesive elements with a continuum response—see "Modeling of an adhesive layer of finite thickness" in "Defining the constitutive response of cohesive elements using a continuum approach," Section 29.5.5 of the Abaqus Analysis User's Manual, or gasket elements with a material response—see "Defining the gasket behavior using a material model," Section 29.6.5 of the Abaqus Analysis User's Manual).

Set TYPE=TRACTION to define effective thickness-direction gasket properties. This option is supported only for gasket elements whose behavior is modeled directly using a gasket behavior model ("Defining the gasket behavior directly using a gasket behavior model," Section 29.6.6 of the Abaqus Analysis User's Manual).

Optional parameters when test data are given to define time domain viscoelasticity with TIME=CREEP TEST DATA or TIME=RELAXATION TEST DATA or when test data are given to define frequency domain viscoelasticity with FREQUENCY=PRONY, FREQUENCY=CREEP TEST DATA, or FREQUENCY=RELAXATION TEST DATA:

ERRTOL

Set this parameter equal to the allowable average root-mean-square error of the data points in the least-squares fit. The default is 0.01 (1%).

NMAX

Set this parameter equal to the maximum number of terms N in the Prony series. Abaqus will perform the least-squares fit from $N = 1$ to $N =$ NMAX until convergence is achieved for the lowest N with respect to ERRTOL. The default and maximum value is 13.

Data line to define continuum material properties for FREQUENCY=FORMULA:

First (and only) line:

1. Real part of g_1^*. $(g^*(\omega) = g_1^* f^{-a})$
2. Imaginary part of g_1^*.
3. Value of *a*.
4. Real part of k_1^*. $(k^*(\omega) = k_1^* f^{-b})$ If the material is incompressible, this value is ignored.
5. Imaginary part of k_1^*. If the material is incompressible, this value is ignored.
6. Value of *b*. If the material is incompressible, this value is ignored.

Data lines to define continuum material properties for FREQUENCY=TABULAR without the PRELOAD parameter or for TIME=FREQUENCY DATA:

First line:

1. Real part of ωg^*. $(\omega \Re(g^*) = G_\ell / G_\infty)$
2. Imaginary part of ωg^*. $(\omega \Im(g*) = 1 - G_s / G_\infty)$
3. Real part of ωk^*. $(\omega \Re(k^*) = \mathrm{K}_\ell / \mathrm{K}_\infty)$ If the material is incompressible, this value is ignored.
4. Imaginary part of ωk^*. $(\omega \Im(k^*) = 1 - \mathrm{K}_s / \mathrm{K}_\infty)$ If the material is incompressible, this value is ignored.
5. Frequency, *f*, in cycles per time.

Repeat this data line as often as necessary to define the dissipative part of the material behavior.

Data lines to define continuum material properties for FREQUENCY=TABULAR, PRELOAD=UNIAXIAL:

First line:

1. Uniaxial loss modulus.
2. Uniaxial storage modulus.

3. Frequency, *f*, in cycles per time.
4. Uniaxial nominal strain (defines the level of uniaxial preload).

Repeat this data line as often as necessary to define the uniaxial loss and storage moduli as functions of frequency and preload.

Data lines to define continuum material properties for FREQUENCY=TABULAR, PRELOAD=VOLUMETRIC:

First line:

1. Bulk loss modulus.
2. Bulk storage modulus.
3. Frequency, *f*, in cycles per time.
4. Volume ratio, J (current volume/original volume; defines the level of volumetric preload).

Repeat this data line as often as necessary to define the bulk loss and storage moduli as functions of frequency and preload.

Data lines to specify continuum material properties with the Prony series parameters directly using TIME=PRONY or FREQUENCY=PRONY:

First line:

1. $\bar{g}_1^P$, the modulus ratio in the first term in the Prony series expansion of the shear relaxation modulus.
2. $\bar{k}_1^P$, the modulus ratio in the first term in the Prony series expansion of the bulk relaxation modulus.
3. τ_1, the relaxation time for the first term in the Prony series expansion.

Repeat this data line as often as necessary to define the second, third, etc. terms in the Prony series. There is no restriction on the number of terms in the Prony series.

To specify viscoelastic behavior via test data:

No data lines are used with this option when either TIME=CREEP TEST DATA or TIME=RELAXATION TEST DATA is specified. The test data are given by the *SHEAR TEST DATA and the *VOLUMETRIC TEST DATA options or by the *COMBINED TEST DATA option.

Data lines to define effective thickness-direction gasket properties for PRELOAD=UNIAXIAL:

First line:

1. Effective thickness-direction loss modulus.
2. Effective thickness-direction storage modulus.
3. Frequency, *f*, in cycles per time.

4. Closure (defines the level of preload).

Repeat this data line as often as necessary to define the effective thickness-direction gasket loss and storage moduli as functions of frequency and preload.

Data lines to define effective thickness-direction gasket properties if PRELOAD=UNIAXIAL is not included:

First line:

1. Real part of ωk^*. ($\omega \Re(k^*) = k_\ell / k_\infty$), where k^* represents the complex effective thickness direction dynamic stiffness.
2. Imaginary part of ωk^*. ($\omega \Im(k*) = 1 - k_s / k_\infty$), where k^* represents the complex effective thickness direction dynamic stiffness.
3. Frequency, *f*, in cycles per time.

Repeat this data line as often as necessary to define the normalized effective thickness-direction gasket loss and storage moduli as functions of frequency.

21.6 *VISCOUS: Specify viscous material properties for the two-layer viscoplastic model.

This option is used to define the viscous properties for the two-layer viscoplastic material model. It must be used in conjunction with the *ELASTIC and *PLASTIC options.

Products: Abaqus/Standard Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Two-layer viscoplasticity," Section 20.2.11 of the Abaqus Analysis User's Manual
- *ELASTIC
- *PLASTIC
- *POTENTIAL

Optional parameters:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the material properties, in addition to temperature. If this parameter is omitted, it is assumed that the material properties depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

LAW

Set LAW=STRAIN (default) to choose a strain-hardening power law.
Set LAW=TIME to choose a time-hardening power law.
Set LAW=USER to input the creep law using user subroutine **CREEP**.

Data lines for LAW=TIME or LAW=STRAIN:

First line:

1. A. (Units of $\mathrm{F}^{-n}\mathrm{L}^{2n}\mathrm{T}^{-1-m}$.)
2. n.
3. m.
4. f.

5. Temperature.
6. First field variable.
7. Second field variable.
8. Third field variable.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than three):

1. Fourth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the viscous constants on temperature and other predefined field variable.

Data lines for LAW=USER:

First line:

1. *f*.
2. Temperature.
3. First field variable.
4. Second field variable.
5. Etc. up to seven field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than seven):

1. Eighth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of the viscous constants on temperature and other predefined field variables.

21.7 *VOID NUCLEATION: Define the nucleation of voids in a porous material.

This option is used to model the nucleation of voids in a porous material. It can be used only with the *POROUS METAL PLASTICITY option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

References:

- "Porous metal plasticity," Section 20.2.9 of the Abaqus Analysis User's Manual
- *POROUS METAL PLASTICITY

Optional parameter:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies, in addition to temperature, that define the normal distribution of the nucleation strain. If this parameter is omitted, the constants defining the normal distribution may depend only on temperature. See "Specifying field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to define void nucleation:

First line:

1. ε_N, mean value of the nucleation-strain normal distribution.
2. s_N, standard deviation of the nucleation-strain normal distribution.
3. f_N, volume fraction of nucleating voids.
4. Temperature.
5. First field variable.
6. Second field variable.
7. Etc., up to four field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than four):

1. Fifth field variable.

2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the dependence of ε_N, s_N, and f_N on temperature and other predefined field variables.

21.8 *VOLUMETRIC TEST DATA: Provide volumetric test data.

This option can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, or the *VISCOELASTIC option.

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Type: Model data

Level: Model

Abaqus/CAE: Property module

Hyperelastic material model

Volumetric loading test data can be provided by this option to include user-defined material compressibility. Compressibility can alternatively be included by using the POISSON parameter on the *HYPERELASTIC option or, in the case of the Marlow model, by specifying the nominal lateral strain under the *UNIAXIAL TEST DATA option. If none of these options is used to specify volumetric behavior, Abaqus/Standard assumes that the material is incompressible, while Abaqus/Explicit will select a default value for the compressibility. See the discussion under "Compressibility" in "Hyperelastic behavior of rubberlike materials," Section 19.5.1 of the Abaqus Analysis User's Manual, before using this option in Abaqus/Explicit.

References:

- "Hyperelastic behavior of rubberlike materials," Section 19.5.1 of the Abaqus Analysis User's Manual
- *HYPERELASTIC

Optional parameter:

SMOOTH

Include this parameter to apply a smoothing filter to the stress-strain data. If the parameter is omitted, no smoothing is performed.

Set this parameter equal to the number n such that $2n + 1$ is equal to the total number of data points in the moving window through which a cubic polynomial is fit using the least-squares method. n should be larger than 1. The default is SMOOTH=3.

Optional parameter when the *VOLUMETRIC TEST DATA option is used in conjunction with the *HYPERELASTIC, MARLOW option:

DEPENDENCIES

Set this parameter equal to the number of field variable dependencies included in the definition of the test data. If this parameter is omitted, it is assumed that the test data depend only on temperature. See "Using the DEPENDENCIES parameter to define field variable dependence" in "Material data definition," Section 18.1.2 of the Abaqus Analysis User's Manual, for more information.

Data lines to specify volumetric test data for hyperelasticity other than the Marlow model (the volume ratios must be arranged in either ascending or descending order if the SMOOTH parameter is used):

First line:

1. Pressure, p.
2. Volume ratio, J (current volume/original volume).

Repeat this data line as often as necessary.

Data lines to specify volumetric test data for the Marlow model (the volume ratios must be arranged in descending order if the SMOOTH parameter is used):

First line:

1. Pressure, p.
2. Volume ratio, J.
3. Temperature, θ.
4. First field variable.
5. Second field variable.
6. Etc., up to five field variables.

Subsequent lines (only needed if the DEPENDENCIES parameter has a value greater than five):

1. Sixth field variable.
2. Etc., up to eight field variables per line.

Repeat this set of data lines as often as necessary to define the test data as a function of temperature and other predefined field variables. The volume ratios must be given in descending order.

Hyperfoam material model

References:

- "Hyperelastic behavior in elastomeric foams," Section 19.5.2 of the Abaqus Analysis User's Manual
- *HYPERFOAM

There are no parameters associated with this option.

Data lines to specify volumetric test data for a hyperfoam:

First line:

1. Pressure, p.
2. Volume ratio, J (current volume/original volume).

Repeat this data line as often as necessary.

Viscoelastic material model

References:

- "Time domain viscoelasticity," Section 19.7.1 of the Abaqus Analysis User's Manual
- *VISCOELASTIC

Optional parameter:

VOLINF

To specify creep test data, set this parameter equal to the value of the long-term, normalized volumetric compliance, $j_K(\infty)$.

To specify relaxation test data, set this parameter equal to the value of the long-term, normalized volumetric modulus $k_R(\infty)$.

The volumetric compliance is related to the volumetric modulus by $j_K(\infty) = 1/k_R(\infty)$. The fitting procedure will use this value in the constraint $1 - \sum_{i=1}^{N} \bar{k}_i^P = k_R(\infty)$.

Data lines to specify volumetric creep test data for a viscoelastic material:

First line:

1. Normalized volumetric (bulk) compliance $j_K(t),\ (j_K(t) \geq 1)$.
2. Time $t,\ (t > 0)$.

Repeat this data line as often as necessary to give the compliance-time data.

Data lines to specify volumetric relaxation test data for a viscoelastic material:

First line:

1. Normalized volumetric (bulk) modulus $k_R(t),\ (0 \leq k_R(t) \leq 1)$.
2. Time $t,\ (t > 0)$.

Repeat this data line as often as necessary to give the modulus-time data.

22. W, X, Y, Z

22.1 *WAVE: Define gravity waves for use in immersed structure calculations.

This option is used to define gravity waves for use in applying loads.

Product: Abaqus/Aqua

Type: Model data

Level: Model

Reference:

- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Manual

Optional parameters:

INPUT

Set this parameter equal to the name of the alternate input file containing the data lines for this option. See "Input syntax rules," Section 1.2.1 of the Abaqus Analysis User's Manual, for the syntax of such file names. If this parameter is omitted, it is assumed that the data follow the keyword line.

TYPE

Set TYPE=STOKES (default) to use Stokes fifth-order wave theory.

Set TYPE=AIRY to use Airy (linearized) wave theory.

Set TYPE=GRIDDED to use gridded data to define the fluid particle velocities, accelerations, free surface elevation, and dynamic pressure.

Set TYPE=USER to use user subroutine **UWAVE** to define the fluid particle velocities, accelerations, free surface elevation, and dynamic pressure and to specify when to update the intermediate configuration for stochastic analysis.

Optional parameter for TYPE=AIRY:

WAVE PERIOD

Include this parameter to indicate that the second field in the data line specifies the wave period, τ_N. If this parameter is omitted, the second field in the data line specifies the wavelength, λ_N.

Required parameter for TYPE=GRIDDED:

DATA FILE

Set this parameter equal to the name of the file containing the gridded data. The file must be a sequential, binary-format file containing records in the format described in "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Manual.

Optional parameters for TYPE=GRIDDED:

MINIMUM

Set this parameter equal to the elevation below which point the structure is fully immersed for all time t. If this parameter is omitted, the elevation of the structure is compared against the instantaneous free surface to check for fluid surface penetration.

QUADRATIC

Include this parameter to indicate that quadratic interpolation of the wave data is used to determine information between grid points. If this parameter is omitted, linear interpolation is used.

Optional parameter for TYPE=USER:

STOCHASTIC

Include this parameter to make the intermediate configuration available to user subroutine **UWAVE**. Set this parameter equal to a random number seed for use in stochastic analysis. If this parameter is omitted or is included without a value, a default value of 0.0 is used for the random number seed. This value is passed into user subroutine **UWAVE**. It is not used otherwise by Abaqus/Aqua.

Data line to define Stokes fifth-order waves (TYPE=STOKES):

First (and only) line:

1. Wave height, H.
2. Period of wave, τ.
3. Phase angle, ϕ, in degrees.
4. Direction cosines giving the direction of travel of the wave, d. In three-dimensional cases both x- and y-components are needed; in two-dimensional cases only the x-component is used (in that case +1 means the wave travels in the direction of x increasing, and -1 means a component traveling in the direction of x decreasing).

Data lines to define Airy waves (TYPE=AIRY):

First line:

1. Wave component amplitude, a_N.
2. Wavelength of this component, λ_N (default), or wave period of this component, τ_N (if the parameter WAVE PERIOD is used).
3. Phase angle of this component, ϕ_N, in degrees.
4. x-direction cosine defining the direction of the vector d_N (the direction of travel for this wave component). In three-dimensional cases both x- and y-components are needed; in two-dimensional cases only the x-component is used (in that case +1.0 means the wave travels in the direction of x increasing, and -1.0 means a component traveling in the direction of x decreasing).

5. y-direction cosine defining the direction of the vector d_N (the direction of travel for this wave component). This component is not needed in two-dimensional cases.

Repeat this data line as often as necessary to define multiple wave trains; one line per wave component.

Data line to define gridded wave data (TYPE=GRIDDED):

First (and only) line:

1. x-coordinate of the origin of the wave data grid.
2. y-coordinate of the origin of the wave data grid.
3. z-coordinate of the origin of the wave data grid.
4. Direction cosine between the local x-axis of the wave data grid and the global x-axis. (Plus or minus one for two-dimensional analysis.)
5. Direction cosine between the local x-axis of the wave data grid and the global y-axis. (Zero for two-dimensional analysis.)

Data line to define frequency versus wave amplitude data for stochastic user wave theory (TYPE=USER):

First line:

1. Frequency (in cycles per time, for stochastic analysis only).
2. Wave amplitude.

Repeat this data line as often as necessary to define the wave spectrum. These data pairs are passed into user subroutine **UWAVE**. They are not used otherwise by Abaqus/Aqua.

22.2 *WIND: Define wind velocity profile for wind loading.

This option is used to define a wind velocity profile for use in applying loads.

Product: Abaqus/Aqua

Type: Model data

Level: Model

Reference:

- "Abaqus/Aqua analysis," Section 6.11.1 of the Abaqus Analysis User's Manual

There are no parameters associated with this option.

Data line to define the wind velocity profile:

First (and only) line:

1. Air density, ρ_a.
2. Reference height for wind profile, z_0.
3. Local x-direction velocity constant, c_x (default value 1.0).
4. Local y-direction velocity constant, c_y (default value 1.0).
5. x-direction cosine defining the direction of the unit vector $\hat{\mathbf{i}}$ (the wind local x-axis) with respect to the global coordinate system. In three-dimensional cases both x- and y-components are needed; in two-dimensional cases only the x-component is used.
6. y-direction cosine defining the direction of the unit vector $\hat{\mathbf{i}}$ (the wind local x-axis) with respect to the global coordinate system. This component is not needed in two-dimensional cases.
7. α (default value 1/7).

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis; multiphysics solutions for insight into challenging engineering problems; and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Vélizy, France, SIMULIA provides sales, services, and support through a global network of regional offices and distributors. For more information, visit www.simulia.com.

About Dassault Systèmes

As a world leader in 3D and Product Lifecycle Management (PLM) solutions, Dassault Systèmes brings value to more than 100,000 customers in 80 countries. A pioneer in the 3D software market since 1981, Dassault Systèmes develops and markets PLM application software and services that support industrial processes and provide a 3D vision of the entire lifecycle of products from conception to maintenance to recycling. The Dassault Systèmes portfolio consists of CATIA for designing the virtual product, SolidWorks for 3D mechanical design, DELMIA for virtual production, SIMULIA for virtual testing, ENOVIA for global collaborative lifecycle management, and 3DVIA for online 3D lifelike experiences. Dassault Systèmes' shares are listed on Euronext Paris (#13065, DSY.PA) and Dassault Systèmes' ADRs may be traded on the US Over-The-Counter (OTC) market (DASTY). For more information, visit www.3ds.com.

Abaqus, the 3DS logo, SIMULIA, CATIA, SolidWorks, DELMIA, ENOVIA, 3DVIA, and Unified FEA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries. Other company, product, and service names may be trademarks or service marks of their respective owners.

© Dassault Systèmes, 2010

DASSAULT SYSTEMES